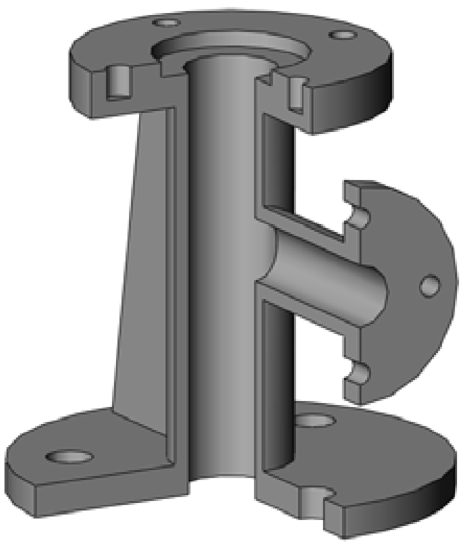
MANUAL
FACULTEIT CONSTRUERENDE TECHNISCHE WETENSCHAPPEN
VAKGROEP ONTWERP, PRODUCTIE EN MANAGEMENT

CAD/CAM HANDLEIDING SOLIDWORKS DEEL I



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CAD/CAM
Handleiding SolidWorks

Deel I

Universiteit Twente
Faculteit Construerende Technische Wetenschappen
Vakgroep Ontwerp, Productie en Management

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Inhoudsopgave

Inleiding

In dit hoofdstuk staat beschreven wat een 3D tekenprogramma nu eigenlijk is, wat het programma SolidWorks daarmee te maken heeft en wat je er mee kunt. Ook staat er beschreven wat het doel van deze handleiding is.

Wat is Solid Modelling?

Onder modelleren verstaan we:
- het maken van een afbeelding van de werkelijkheid,
- het vormgeven of boetseren (Engels shape).

In de techniek zijn vorm en afmetingen twee belangrijke kenmerken waar dagelijks mee wordt gewerkt. Een as heeft een bepaalde diameter en lengte, een gat een bepaalde diameter, positie en diepte. Het modelleren waar in deze handleiding over gesproken wordt (Geometric Modeling), is een activiteit waarbij de vorm en afmetingen van een object worden vastgelegd in wiskundige formuleringen. Van de gebruikte wiskundige formuleringen zul je overigens als gebruiker van het programma SolidWorks over het algemeen weinig merken.

Vroeger was de enige mogelijkheid om geometrieën vast te leggen het maken van handschetsen of tekeningen. Hierbij wordt in twee of drie aanzichten en/of doorsneden aangegeven hoe een bepaald onderdeel er uit ziet. Dit is een tweedimensionale techniek, aangezien er 2D tekeningen worden gemaakt van het onderdeel. In dit soort tekeningen mist dus altijd de derde dimensie, bijvoorbeeld de diepte of hoogte van een gat of onderdeel. De tekeningen zijn meestal orthogonaal geprojecteerde aanzichten, dus loodrecht op het tekenvlak.

Naast het maken van handschetsen en tekeningen is het tegenwoordig ook mogelijk om driedimensionale modellen te maken in speciale 3D modelleerprogramma's. Deze programma's bieden de mogelijkheid aan gebruikers om met behulp van verschillende technieken een 3D model op een computer op te bouwen. Voordelen hiervan zijn onder andere dat driedimensionale modellen het begrip bij technici van de opbouw van onderdelen en samenstellingen vergroten. Bovendien krijgen onderdelen steeds complexere vormen. Denk hierbij bijvoorbeeld aan de driedimensionaal gekromde oppervlakken van moderne scheerapparaten.

Na het driedimensionaal modelleren van onderdelen, kunnen door de 3D modelleerprogramma's vrij eenvoudig tweedimensionale tekeningen worden gegenereerd van door de gebruiker gedefinieerde aanzichten. De gebruiker hoeft dus niet meer zelf tweedimensionaal te werken. Afmetingen worden gerelateerd aan het driedimensionale model. In de tekeningen zullen dus niet snel foute afmetingen kunnen worden afgelezen, tenzij deze natuurlijk al in het driedimensionale model aanwezig zijn. Ook kunnen verschillende onderdelen in elkaar worden gepast in zogenaamde 'Assemblies'. Dit biedt een controle mogelijkheid voor de maakbaarheid van bepaalde samenstellingen.

Wat is SolidWorks?

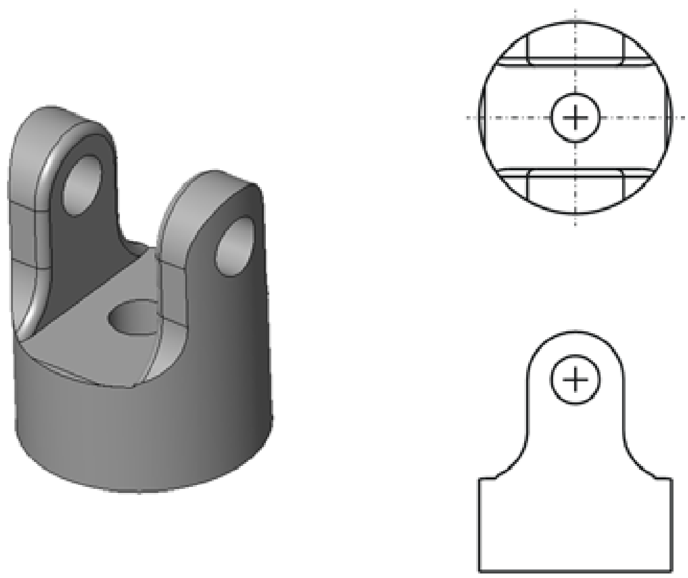
SolidWorks is een 3D modelleerprogramma waarin je 3D onderdelen kunt creëren. Je kunt deze 3D onderdelen gebruiken voor het genereren van 2D tekeningen en 3D samenstellingen.
De bematingen zijn de drijvende krachten achter het model. Je kunt bematingen opgeven en geometrische relaties tussen elementen invoeren. Verander je de maten, dan verander je ook de grootte en de vorm van het onderdeel, terwijl het ontwerpdoel behouden blijft.

Een SolidWorks 3D model bestaat uit onderdelen, samenstellingen en tekeningen. Onderdelen, samenstellingen en tekeningen zijn verschillende afbeeldingen van hetzelfde model. Elke verandering die gemaakt wordt in één van de aanzichten wordt automatisch in alle andere afbeeldingen aangepast.

Onderdelen worden aangemaakt met behulp van verschillende features (eigenschappen). Features zijn de vormen (bosses, cuts, holes) en operaties (fillets, chamfers, shells, enzovoort) die gecombineerd worden bij het maken van onderdelen.

De meeste features worden gemaakt vanuit een sketch (schets). Een sketch is een 2D profiel of een doorsnede. Sketches kunnen bijvoorbeeld worden geëxtrudeerd of er kan een omwentelingslichaam van worden gemaakt.

SolidWorks is het modelleer programma dat bij de faculteit CTW aan de Universiteit Twente is gekozen als standaard gereedschap bij alle ontwerp activiteiten voor het CAD/CAM onderwijs.



3D en 2D weergave van een onderdeel

Doel

Het doel van deze handleiding is om op een overzichtelijke manier de functies van het programma SolidWorks te beschrijven. Als nieuwe gebruiker van het programma SolidWorks zou je, na het lezen van deze handleiding en het doen van de oefeningen die erin staan, in staat moeten zijn om zelf een part, assembly en/of drawing te creëren.

Structuur van deze handleiding

De handleiding bestaat uit zes hoofdstukken. De hoofdstukken zijn opgedeeld in paragrafen en bestaan uit beschrijvende tekst en opdrachten die de lezer kan uitvoeren om het begrip van het programma te verhogen.

Het eerste hoofdstuk bestaat uit de inleiding. Het brengt de lezer wat basisbegrippen en gebruiksvaardigheden voor het programma bij. In hoofdstuk 2 wordt de gebruiker door middel van opdrachten geleerd hoe in SolidWorks kan worden gemodelleerd. Welke features kunnen waarvoor worden gebruikt en welke opties kunnen waar worden gevonden. Hoofdstuk 3 behandelt vervolgens het samenvoegen van verschillende parts in een assembly, waarna hoofdstuk 4 beschrijft hoe van de parts een tweedimensionale tekening kan worden gemaakt. Enkele speciale onderwerpen, zoals het modelleren van plaatwerk, komen aan bod in hoofdstuk 5. Hoofdstuk 6 beschrijft de tool "Hole Wizard", waarmee gemakkelijk allerlei verschillende soorten gaten gemaakt kunnen worden. Hoofdstuk 7 gaat in op het renderen van producten.

Hoofdstuk 1:
Basis

In dit hoofdstuk worden de basisfuncties van het programma SolidWorks uitgelegd. Er staat onder andere in hoe je het programma moet starten, de programma-omgeving wordt uitgelegd en er wordt uitgelegd hoe je je document op verschillende manieren kunt weergeven.

Starten

1. Klik op de **Start** knop van de Windows taskbar (taakbalk).

2. Klik **All Programs,**

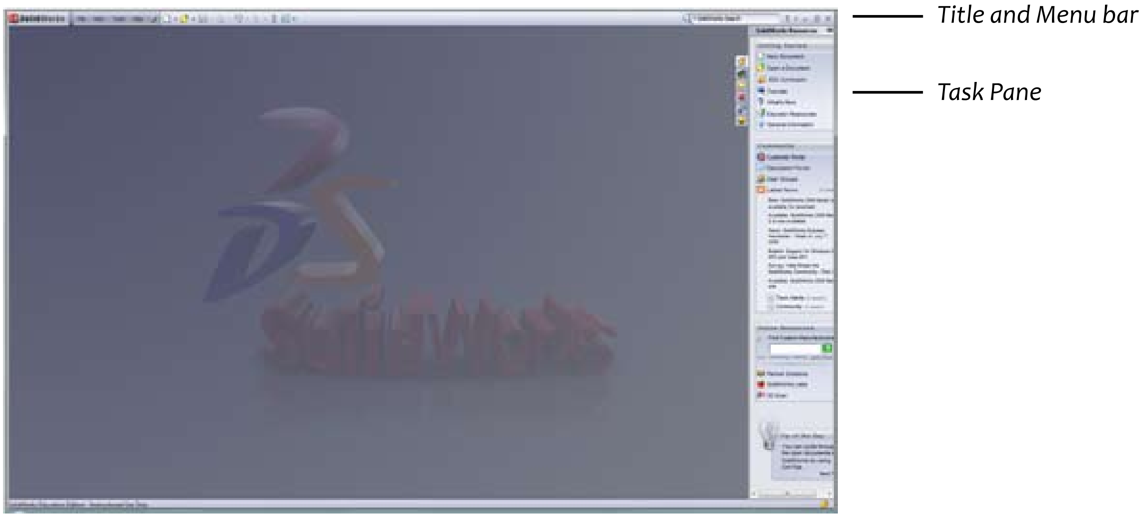
3. **SolidWorks 2010,**

4. **SolidWorks 2010** (32- of 64-bits)**.**

Het SolidWorks main window (hoofdvenster) verschijnt en het 'Welcome to SolidWorks 2010' scherm opent.

Programma-omgeving verkennen

Na het starten van het programma krijg je onderstaand scherm te zien.



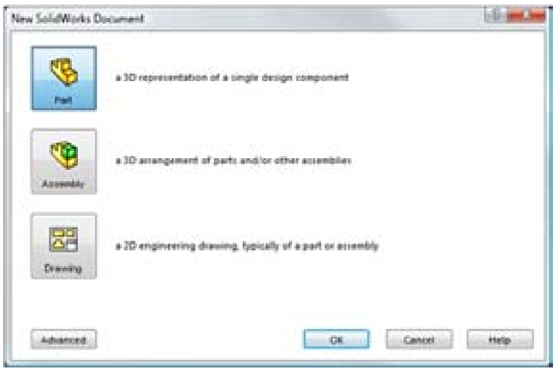
De Title and Menu bar (titel- en menubalk) laat zien welk document geopend is, en zal een aantal menu's laten zien zoals 'File', 'View' enzovoort. Deze zal ook de opties weergeven voor je document als je een document geopend hebt.

Controleren van de instellingen

Om ervoor te zorgen dat hetgeen op je scherm verschijnt, zoveel mogelijk overeenkomt met de afbeeldingen uit de handleiding, kun je de instellingen gelijk maken aan die van de schrijver van deze handleiding. Controleer daarom voor je begint, of de SolidWorks instellingen overeenkomen met de waarden die in deze handleiding worden gebruikt.

5. Klik op de icoon **New** , of klik **File, New...**

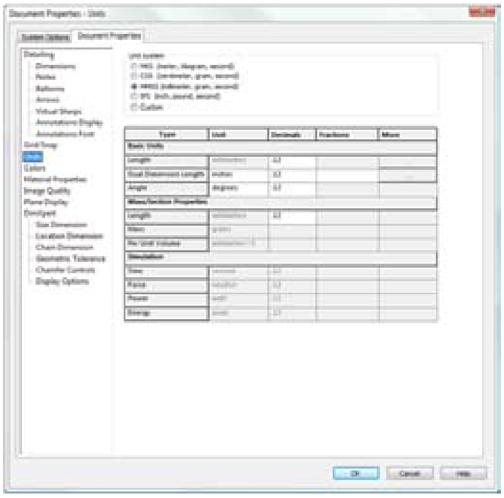
6. Selecteer **Part** en klik op **OK** (of dubbelklik op **Part**).



7. Klik op de **Tools** knop op de Menu bar.

8. Klik **Options...**

   Het Options scherm verschijnt.



9. Selecteer in de linker boom van het tabblad **Document Properties** de groep **Units**.

10. Controleer of **Unit system** op **MMGS(millimeter,gram,second)** staat.

    Indien het Unit System niet op MMGS staat, moet je een nieuwe ‘template‘ aanmaken. In een template kun je de instellingen voor je document bewaren. Door een nieuw template aan te maken met de juiste instellingen, weet je zeker dat je bij elk nieuw te ontwerpen onderdeel steeds met de juiste instellingen begint.

    de onderstaande stappen moet je alleen uitvoeren indien het Unit System **niet** op MMGS staat.

11. Zet **Unit system** op **MMGS(millimeter,gram,second)**.

12. Zet alle **Decimals** in de tabel (die je kan aanpassen) op **.12**.

13. Klik op **OK**.

14. Klik op **File** in de Menu bar en vervolgens op **Save as...**

15. Selecteer onder **Save as type** de optie **Part Templates (*.prtdot)**.

    Als je dit hebt geselecteerd zal de directory waarin het template wordt opgeslagen automatisch verspringen naar 'SolidWorks Data\templates'.

16. Geef je template de naam **Part.prtdot** en klik **Save**. bij de vraag of je het origineel wilt vervangen zeg je **ja**.

Help

Als je vragen hebt terwijl je met het programma SolidWorks werkt, zijn er verschillende methoden om antwoorden op je vragen te vinden:

- Klik op de **Help** knop op de Menu bar en klik dan **SolidWorks Help Topics**. De help heeft een **Index** en een **Search** functie.
- Voor handige tips, klik op de **Help** knop op de Menu bar en klik dan **Quick tips**.
- Klik in een dialog van de PropertyManager op  , of druk op **F1** voor een uitleg van de betreffende functie.
- Voor **Tooltips**, een korte omschrijving van de knoppen op de toolbar, ga je met de cursor van je muis op de knop staan. Een moment later verschijnt dan de tooltip.
- Wanneer je je muis over knoppen of menu's beweegt, verschijnt er in de Status bar, die onderin het scherm te vinden is, een korte omschrijving van de functie hiervan.

Voor meer informatie en het laatste nieuws over het programma en het bedrijf SolidWorks, kun je de SolidWorks website, http://www.solidworks.com bezoeken. Je kunt ook op de **Help** knop op de Menu bar klikken en dan **About SolidWorks, Connect** selecteren.

Weergave-mogelijkheden

In dit gedeelte worden de visualisatiemogelijkheden binnen SolidWorks weergegeven. Er wordt beschreven hoe je gebruik kunt maken van de **Zoom Tools**, de verschillende **Display Styles**, het roteren of verplaatsen van modellen en hoe je gebruik kunt maken van meerdere aanzichten. Voor het gebruik van de weergavemogelijkheden kun je de **View (Heads-up) toolbar** gebruiken. deze bevindt zich bovenaan het scherm en ziet er zo uit:

voor het oefenen met de weergave-mogelijkheden gebruiken we een voorbeeld-part.

1. Open het part '**bracket.sldprt**' in de map '**Chapter 1**' door op het **Open** icoon  of op **File, Open...** te klikken.

2. Test de hieronder genoemde opdrachten op het part.

Aanpassen van de View toolbar

Op dit moment staan nog niet alle Tools in de toolbar die wij hier zullen gebruiken. Ook staan er sommige die we niet zullen gebruiken. Daarom gaan we eerst de toolbar aanpassen.

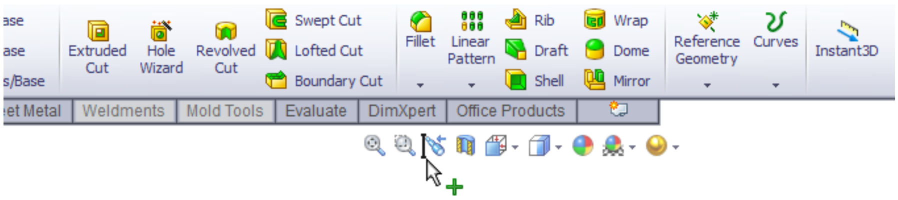
1. Klik op **Tools, Cusomize...**

2. Klik op het tabblad **Commands**.

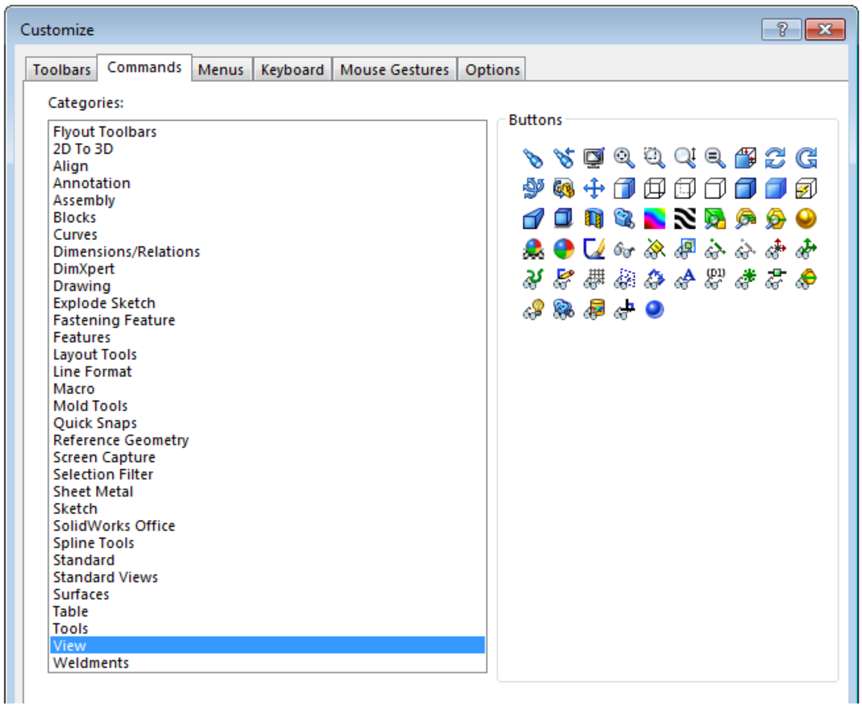
3. Klik op de categorie **View**.

4. verplaats het Customize-menu zodanig, dat je de incoontjes in het menu kunt zien en tegelijkertijd de toolbar.

5. sleep het **Rotate View** icoon naar de **view toolbar**.



Op het moment van verslepen zie je een **groen plusje** verschijnen. dit geeft aan dat je het icoon toevoegt. Daarnaast verschijnt er een zwarte verticale streep. Hiermee kun je de positie van het icoon op de toolbar bepalen.



6. versleep ook het **Pan** icoon naar de toolbar.

Naast het toevoegen, kun je ook iconen verwijderen. Dit kan door op een icoon te klikken, de knop ingedrukt te houden en het icoon van de toolbar af slepen. Op het moment dat je met je cursor van de toolbar af bent, zie je een **rood kruisje** verschijnen. Dit betekent dat het icoon verdwijnt van de toolbar als je hem nu loslaat.

7. Sleep de volgende iconen van de toolbar af: **Previous View** , **Hide/Show Items** , **Apply Scene** en **View Settings** .

De toolbar ziet er nu als het goed is zo uit:

het kan wel zijn dat je een andere volgorde hebt in de iconen op de toolbar.

Gebruik maken van de Zoom Tools

Er zijn twee zoom tools waarmee de afbeelding op het scherm kan worden verkleind en vergroot.

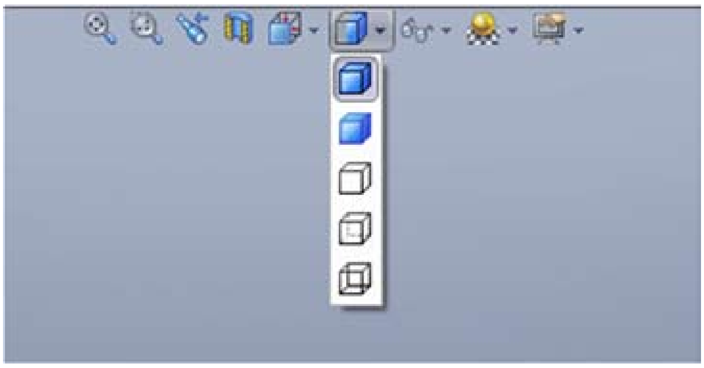
- **Zoom to Fit** past het onderdeel zo groot mogelijk in het window.
- **Zoom to Area** vergroot het gebied binnen een door de gebruiker gedefinieerde rechthoek tot het scherm gevuld is.
- Scrollen met het **scroll wiel** van je muis zoomt in of uit op het onderdeel. Het scherm zoomt hierbij in op het punt waar je cursor staat.

8. gebruik **Zoom to Area** om in te zoomen op een deel van het part 'bracket'.

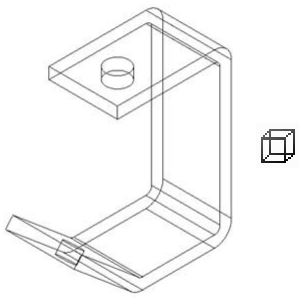
9. gebruik **Zoom to Fit** om weer het volledige model te zien.

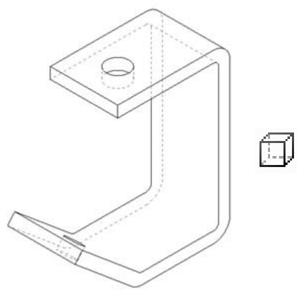
10. Als je een muis met scroll wiel hebt, gebruik deze om het effect er van te zien.

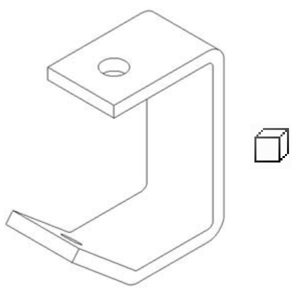
Weergeven in verschillende Display Styles

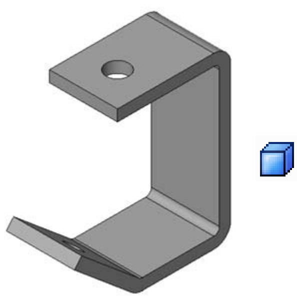


Klik op om het onderdeel weer te geven als **Wireframe**, klik op om het onderdeel te laten verschijnen als **Hidden Lines Visible** (wat de verborgen lijnen in een andere kleur of font laat verschijnen), klik op en op om het voorwerp als **Hidden Lines Removed** respectievelijk **Shaded with edges** te zien. De laatste optie is **Shaded** , dus zonder omlijning. Door een optie te selecteren uit de **View, Display** dialog kun je ook de manier van weergave wijzigen.









De vast ingestelde stand voor onderdelen en samenstellingen is **Shaded with edges**, dit kan natuurlijk op ieder moment worden gewijzigd. Sommige afbeeldingen in deze handleiding zijn weergegeven in de **Hidden Lines Removed** stand, dit om een duidelijke afbeelding in de handleiding te verkrijgen. Het kan daardoor voorkomen dat wat op het scherm te zien is niet geheel overeen komt met de illustraties in deze handleiding.

11. Klik de verschillende weergave opties om de beurt aan en bekijk het resultaat.

Het onderdeel roteren

Er zijn een aantal verschillende mogelijkheden om het onderdeel te roteren:

12. De meest makkelijke methode om het onderdeel te roteren is het gebruiken van het **scroll wiel** van je muis. Heb je een muis met scroll wiel, dan kun je het scroll wiel indrukken en de cursor over het beeld slepen.

13. Gebruik de pijltjes toetsen om het onderdeel in stapjes te laten roteren. De grootte van deze stapjes wordt vastgelegd in **View Rotation, Arrow Keys** van het **System Options** tabblad van de **Options** dialog (onder **Tools, Options...**).

14. Verdraai het onderdeel staploos. klik daarvoor op **Rotate View** uit de View toolbar of **View, Modify, Rotate**. Vervolgens kunt je het onderdeel verdraaien door met de linker muisknop te slepen.

15. Het onderdeel laten draaien om een rand of een vertex: klik op **Rotate View** , dubbelklik op een rand of vertex en sleep de muis.

De afbeelding verplaatsen

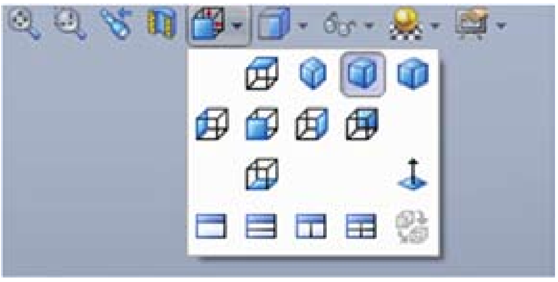
16. Klik op **Pan** uit de View toolbar, of **View, Modify, Pan,** klik en sleep het onderdeel over het scherm.

17. Houd de **Ctrl**-toets ingedrukt en druk op de pijltjes toetsen om de afbeelding over het scherm te verplaatsen.

18. Gebruik de **Scroll Bars** onder en rechts in het scherm om naar een ander deel van het scherm te gaan.

Het View Orientation dialog

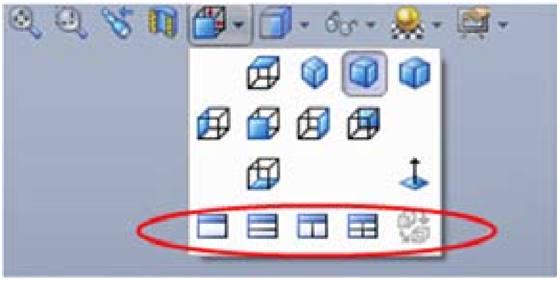
De **View Orientation** dialog bepaalt de oriëntatie van het onderdeel of de assembly ten opzichte van de gebruiker. Er zijn verschillende views (aanzichten) die worden aangeduid met een naam. Er zijn 10 standaard aanzichten gedefinieerd die via het **View Orientation** dialog en de **Standard Views** toolbar bereikbaar zijn. Verder is er een knop voor een isometrische projectie en (heel handig) een knop waarmee het schetsvlak evenwijdig aan het beeldscherm wordt gemaakt.



19. Selecteer een vlak van het part ‘bracket’.

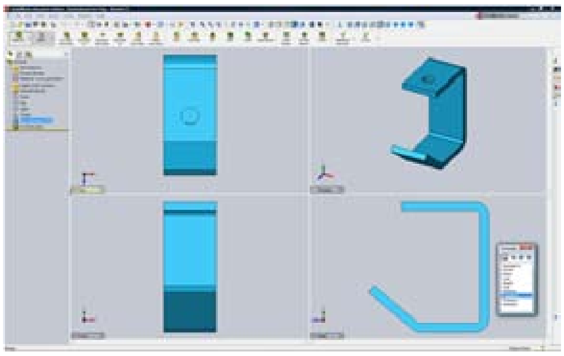
20. Klik **Normal to** in het **View Orientation** dialog

Weergave van meerdere aanzichten



Er kunnen maximaal vier aanzichten van een onderdeel tegelijk op het beeldscherm worden weergegeven. Dit is vooral handig wanneer je verschillende features wilt selecteren van tegenover elkaar liggende vlakken, of als je het effect van een actie van verschillende zijden van het model tegelijk wilt zien. Wanneer je een feature in één aanzicht selecteert, is deze feature in alle aanzichten geselecteerd.

21. Klik op het **Four View** in het **View Orientation** dialog . Je krijgt het volgende resultaat:



Online Handleiding

Wanneer je merkt dat deze handleiding (nog) te moeilijk voor je is, kun je extra informatie over de concepten van het programma vinden in hoofdstuk 1 van de **Online Tutorial**. Deze is online (op je computersysteem) aanwezig. Hij kan bekeken worden door de volgende handelingen uit te voeren:

1. Klik op **Help, Solidworks Tutorials** in de Menu bar.

Hierin vindt je onder andere:

- Terminologie.
- Window eigenschappen, zoals toolbars, menus en views.
- Eenvoudige grafische operaties, zoals het selecteren en het bewegen van objecten.
- De FeatureManager design tree.

Hoofdstuk 2:
Part Modelling

In dit hoofdstuk wordt beschreven wat een part is en hoe je deze kunt maken. Er wordt verteld welke features je hierbij kunt gebruiken. Aan de hand van voorbeelden wordt duidelijk gemaakt hoe je deze features kunt gebruiken.

Wat is een Part?

Part is engels voor onderdeel. Onderdelen worden aangemaakt met behulp van verschillende features (eigenschappen). Features zijn de vormen (bosses, cuts, holes) en operaties (fillets, chamfers, shells, enzovoort) die gecombineerd worden bij het maken van onderdelen.

De meeste features worden gemaakt vanuit een sketch (schets). Een sketch is een 2D profiel of een doorsnede. Sketches kunnen bijvoorbeeld worden geëxtrudeerd of er kan een omwentelingslichaam van worden gemaakt.

Bij het maken van een 3D tekening begin je altijd met het maken van een part. Bestaat je model uit één onderdeel dan zal deze part meteen je eindmodel zijn. Bestaat je model uit meerdere onderdelen, dan zul je van elk onderdeel een part moeten maken, alvorens hier een assembly (samenstelling) van te maken.

Welke features kunnen er gebruikt worden?

Bij het creëren van een part kan van verschillende features gebruik gemaakt worden, die van een tweedimensionale sketch een driedimensionaal model maken.
De verschillende features zijn:

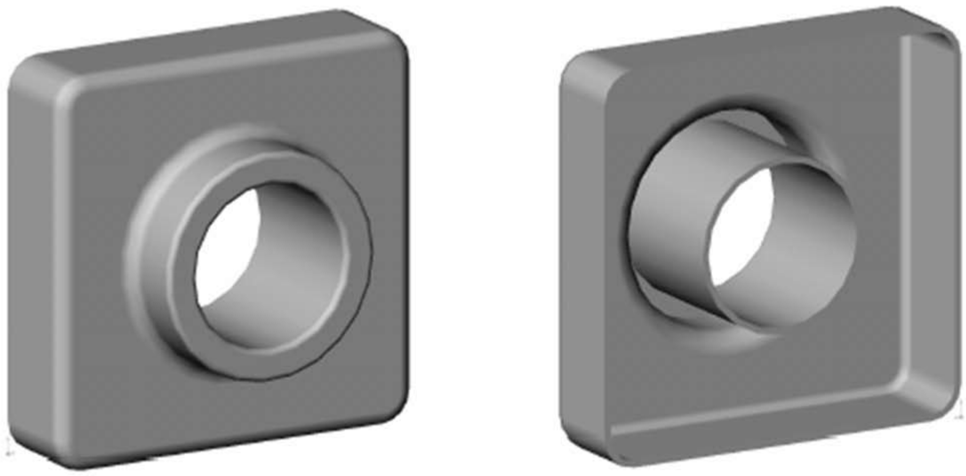
- **Base/Boss** en **Cut**
  Bij elk part is de eerste feature die wordt aangemaakt de base feature. Een boss is een feature die materiaal aan een part toevoegt. Een cut is een feature die materiaal weghaalt van een part of assembly.
- **Extrude, Revolve, Sweep** en **Loft**
  Om een base of boss te creëren gebruik je één van deze vier features. De features kunnen zowel gebruikt worden om materiaal toe te voegen als om materiaal te verwijderen door tussen de opties base, boss, cut of surface te kiezen. Een Extrude creeert een extrusie van een sketch. Revolve creëert een base, boss, cut of surface door een sketch om een centerline te wentelen. Met een Sweep kun je een profiel langs een opgegeven path laten lopen en zo een feature aanmaken. Een Loft tenslotte creëert een feature door het maken van overgangen tussen profielen.

- **Fillet/Round, Chamfer** en **Draft**
  Fillet/Round creëert een afgerond intern of extern vlak op een part. Je kunt hiermee bijvoorbeeld de randen van een vlak afronden. Een Chamfer feature creëert een afgeschuind oppervlak op geselecteerde randen of een knooppunt. Draft laat geselecteerde vlakken in het model onder een bepaalde hoek taps uit- of toelopen. Je kunt hiermee bijvoor beeld een molded part gemakkelijker verwijderen uit de mal.
- **Hole - Simple** en **Hole Wizard**
  Hole creëert verschillende types van gaten in het model. Je kunt een hole op een vlak plaatsen en het een diepte ge- ven. Je kunt de locatie ervan daarna weergeven door het te dimensioneren.
- **Pattern** en **Mirror**
  Deze feature geeft je de mogelijkheid om één of meerdere andere features te kopiëren

Naast deze zijn nog vele andere features mogelijk. Kijk hiervoor naar deel II van de handleiding of Solidworks Help.

Part Basics

Dit gedeelte begeleidt je bij het maken van je eerste SolidWorks model en tekening. Je gaat het onderstaande model maken.

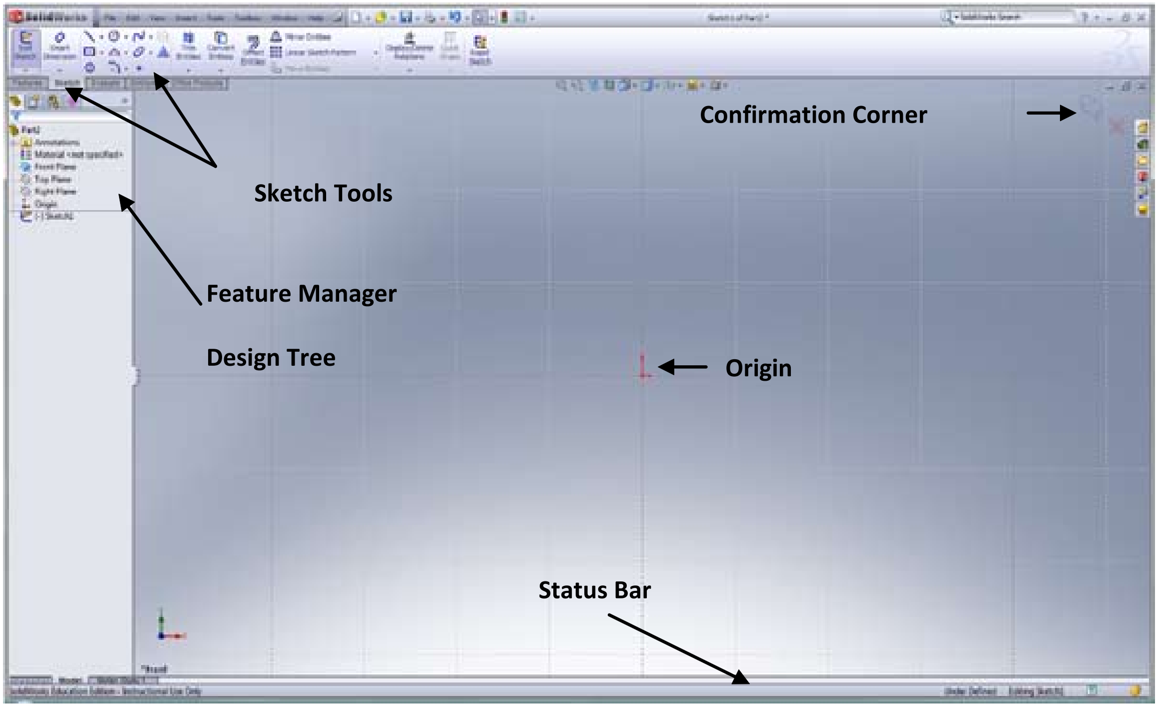


Een nieuw Part document aanmaken

1. Klik op **New Document** of op **File, New...**

2. Selecteer het **Part** icoon en klik op **OK**. Je kunt ook dubbelklikken op het icoon.

   Er verschijnt nu een nieuw part window, zoals te zien is op de volgende pagina.



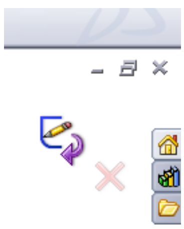
Het schetsen van een rechthoek

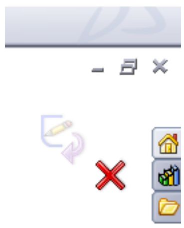
De eerste feature in het onderdeel is een geëxtrudeerd blok vanuit een geschetst rechthoekig profiel. Je begint met het schetsen van de rechthoek.

3. Om een schets te openen, klik je op **Sketch** op de Sketch tab of op **Insert, Sketch** op de Menu bar. Open een sketch in het **Front** plane. Dit vlak is één van de drie standaardvlakken, getoond in de **FeatureManager design tree** en in het tekenvlak.

Merk op dat:

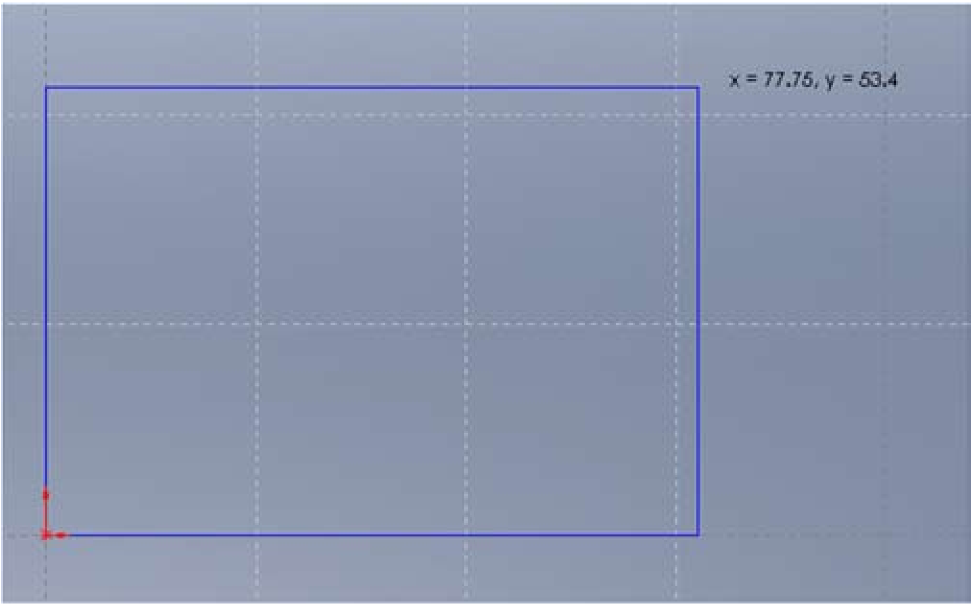
- De Sketch tab weergegeven wordt, als deze nog niet geselecteerd is.
- "**Editing Sketch**" verschijnt in de Status bar onderaan het scherm.
- **Sketch 1** verschijnt in de FeatureManager design tree. Wat hierbij opvalt is dat hij onder de streep staat. hieraan kun je zien dat je op dit moment met deze schets bezig bent.
- In de **rechter bovenhoek** verschijnt de zogenaamde "Confirmation corner." Als een sketch actief is, kome hier twee symbolen te staan. Ze worden vervaagd weergegeven, maar als je met je cursor er overheen gaat, lichten ze op, zoals hiernaast te zien is. Als je op het **Confirm Sketch** symbool uit de eerste afbeelding klikt verlaat je de sketch en worden alle veranderingen opgeslagen. Klik je op **Cancel Sketch,** de rode X uit de tweede afbeelding, dan wordt de sketch verlaten en veranderingen worden genegeerd.





4. Klik op **Rectangle** op de Sketch tab of selecteer in de Menu bar **Tools, Sketch Entity, Rectangle**.

5. Plaats de cursorpunt in de oorsprong en druk op de linker muisknop. Sleep de cursor naar rechts boven, om een rechthoek te maken. Laat de muisknop los om de rechthoek af te maken. Merk op dat tijdens de sleepactie de afmetingen van de rechthoek worden weergegeven bij de cursor.



De twee zijden van de rechthoek die de assen van de oorsprong raken zijn zwart. Omdat de schets is gestart in de oorsprong, is een vertex (punt) van deze twee zijden automatisch gerelateerd aan de oorsprong. Daarmee ligt de positie van deze lijnen vast. De andere twee zijden (en drie vertices) zijn **blauw**, wat er op wijst dat ze nog vrij kunnen bewegen.

6. Klik op één van de blauwe zijden en versleep deze, om de afmetingen van de rechthoek aan te passen. doe hetzelfde met de vertexen.

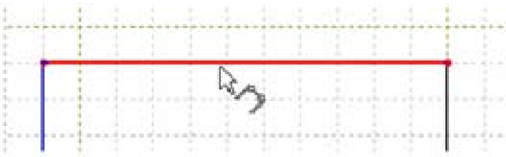
   Merk op dat de positie van de cursor, of de lengte van de zijde wordt weergegeven in de status bar.

Het toevoegen van de bemating

In dit gedeelte leg je de afmetingen van de schets vast door er **dimensions** (bemating) aan toe te voegen.

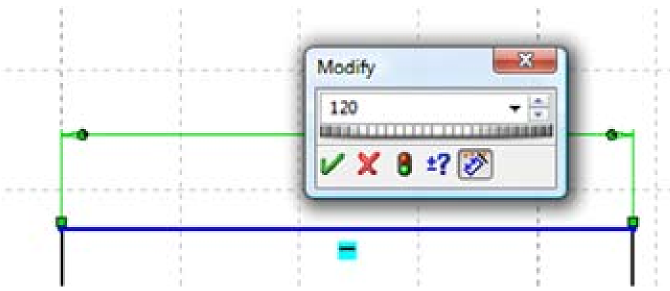
7. Klik op **Smart Dimension**  op de Sketch tab of **Tools, Dimensions, Smart** in de Menu bar.

   Merk op dat de cursorvorm verandert.

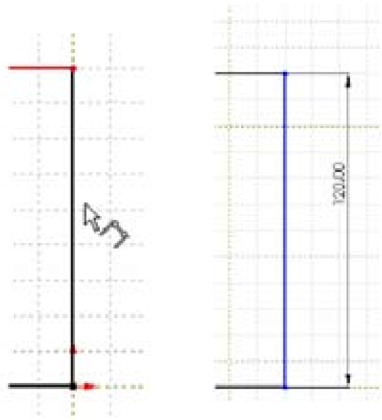


8. Klik op de bovenste rand van de rechthoek en klik op de plek waar je de maatlijn wilt plaatsen.

   De **Modify** dialog verschijnt in het scherm. Hier kunt je bematingen aanpassen. Stel de breedte van het vierkant in op **120 mm** en druk op **OK**  of op Enter om de waarde toe te kennen aan het lijnstuk.



Merk op dat de rechter verticale lijn (en de vertex rechtsonder) veranderen van een blauwe naar een zwarte kleur. Door de dimensionering van de lengte van de bovenste zijde van de rechthoek is de positie van de rechter zijde van de rechthoek gedefinieerd. De bovenste zijde kan nog steeds omhoog en omlaag worden gesleept; dit is te zien aan de blauwe kleur van dit segment, die aanduidt dat het segment nog niet volledig is gedefinieerd.



9. Klik op de rechterrand van de rechthoek en klik op de plek waar de maatlijn moet staan. Stel ook de hoogte van het vierkant in op **120 mm**.

   Nu zijn de rechterrand en de overgebleven vertices **zwart**. Merk op dat in de status bar vermeld staat dat de schets volledig is gedefinieerd.

   Elke SolidWorks schets kan zich in drie mogelijke toestanden bevinden. deze worden altijd onderaan in de status bar vermeld:
   - In een **fully defined sketch** zijn de posities van alle onderdelen volledig vastgelegd door dimensies en/of relaties. In een volledig gedefinieerde schets zijn alle onderdelen **zwart**.
   - In een **underdefined sketch** zijn er buiten de al gegeven dimensies en/of relaties nog aanvullende dimensies en/of re laties nodig om de geometrie volledig te specificeren. In deze staat kun je ondergedefinieerde schetsonderdelen slepen om de schets aan te passen. Een ondergedefinieerd schetsonderdeel is **blauw** gekleurd.
   - In een **overdefined** sketch is een object gespecificeerd door met elkaar conflicterende dimensies en/of relaties. Een overgedefinieerd schetsonderdeel is **geel** gekleurd. Ook wordt in **geel** en **rood** aangegeven dat er een conflict in de maten is.

Het veranderen van de bemating

De dimensies van het blok zijn 120 mm x 120 mm. Mocht je de afmetingen willen veranderen, dan gebruik je de **Select** tool.

10. Gebruik één van de onderstaande methoden om toegang te krijgen tot de **Select** tool:
    - Klik op de **Select** button uit de Sketch tab.
    - Druk de rechtermuisknop in op een leeg gedeelte van de window om het rechtermuismenu te tonen, kies dan **Select**.

    Tip: *Door het rechter-muismenu te gebruiken, kun je SolidWorks efficiënter gebruiken.*

11. Dubbelklik op één van de maatlijnen.

    De **Modify** dialog verschijnt weer.

12. Verander de maat en kijk naar het effect. Zet de maat vervolgens weer op 120 mm.

13. Klik op **Zoom to Fit** uit de View toolbar, of **View, Modify, Zoom to Fit** om de gehele rechthoek volledig en gecentreerd op het scherm te passen.

    Dimensiewaarden kunnen ook meteen bij het plaatsen worden aangepast. Activeer hiertoe de optie **Input dimension value** in de **System Options** dialog. Elke keer wanneer een nieuwe bemating wordt toegevoegd verschijnt de **Modify** dialog, waarin je de waarde kunt aanpassen.

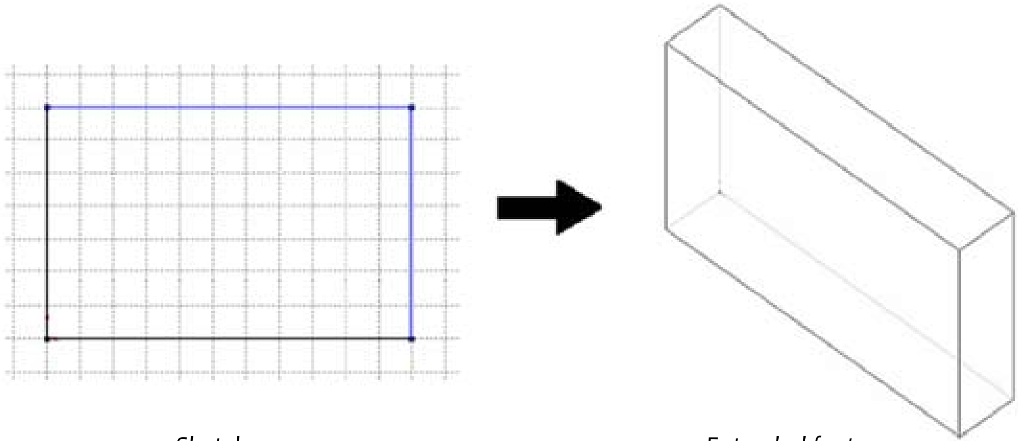
14. Klik **Tools, Options**.

15. Selecteer onder de **System Options** tab de groep **General** en plaats een vink bij **Input dimension value**.

16. Klik op **OK**.

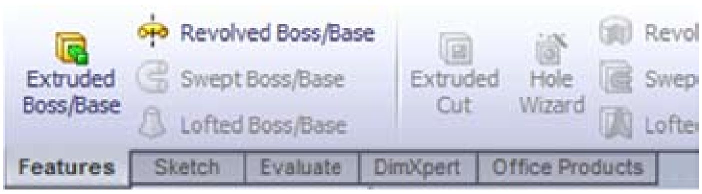
Het extruderen van de Base Feature

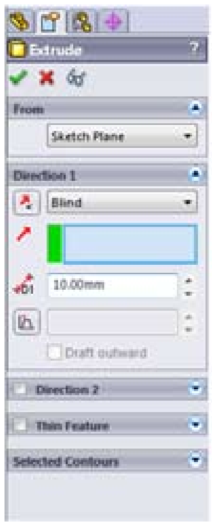
De eerste feature van een onderdeel wordt de base feature genoemd. Het definieert de basisvorm van een nieuw onderdeel. In dit voorbeeld wordt de base feature gemaakt door de geschetste rechthoek te extruderen.



Sketch *Extruded feature*

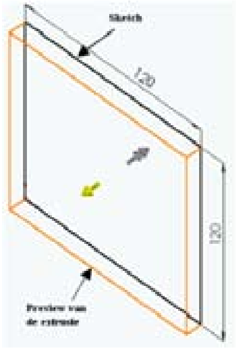
17. Klik **Extrude, Boss/Base**  op de Features tab, of klik **Insert, Base, Extrude**. De **Extrude PropertyManager** verschijnt links en het aanzicht van de schets wijzigt in een isometrisch aanzicht. Merk verder op dat er tooltips verschijnen met de namen van de opties als de cursor in een invulveld wordt geplaatst.

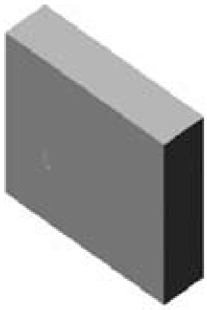




18. Specificeer het type en de diepte van de extrusie:
    - Zorg er voor dat **End Condition** op **Blind** staat.
    - Zet onder **Depth** de diepte op **30 mm**.

Tijdens het veranderen van de diepte, toont SolidWorks een preview van de extrusie. Indien op de optie **Reverse Direction** wordt geklikt, zal de richting van de extrusie worden omgedraaid.

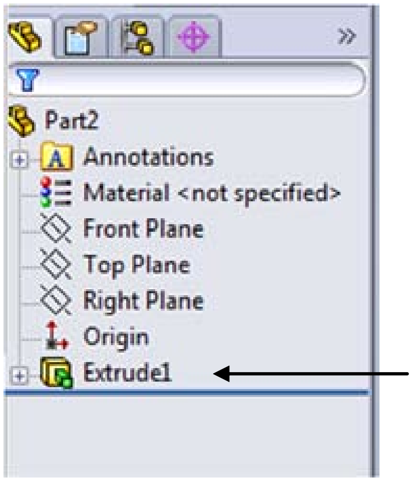




19. Zorg er voor dat **Thin feature** niet aangevinkt is, de sketch zal dan als **Solid feature** geëxtrudeerd worden (dit is standaard).

20. Klik op **OK** in de Extrude PropertyManager of in de rechterbovenhoek van het scherm om de extrusie te genereren.

Merk op dat de nieuwe feature, **Extrude1**, in de FeatureManager design tree staat. Klik op het plus teken **+** naast Extrude1 in de FeatureManager design tree. Hierdoor verschijnt **Sketch1**, die gebruikt is om de feature te extruderen en nu onder de feature Extrude1 staat gerangschikt.

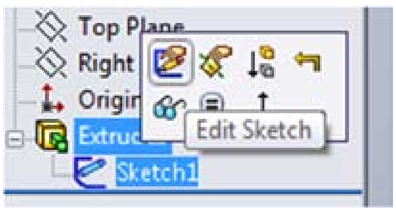


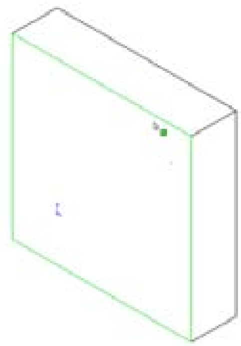
Het Creëren van een Boss

Om nog meer features te creëren op het onderdeel (zoals bosses of cuts) schets je op de vlakken van het onderdeel en extrudeer je vervolgens deze schetsen.

Opmerking: *In SolidWorks schets je maar op één vlak tegelijk en daarmee maak je vervolgens een feature gebaseerd op die ene schets.*

- Om een nieuwe schets te openen selecteer je eerst een vlak waarop wordt geschetst, vervolgens klik je op de **Sketch** tool ,
- Om een schets te sluiten, klik je op **Exit Sketch** op de Sketch tab, op **Confirm** of **Cancel Sketch** in de rechterbovenhoek of je selecteert **Exit Sketch** uit het rechtermuismenu.
- Om een schets aan te passen waar je al eens aan hebt gewerkt, kun je met de linkermuisknop op de feature klikken dat met die schets is gemaakt, of je klikt op de schetsnaam in de FeatureManager design tree. Vervolgens selecteer je **Edit Sketch** uit het popup-menu.





Voor dit voorbeeld wordt een schets gemaakt op het voorvlak van het blok.

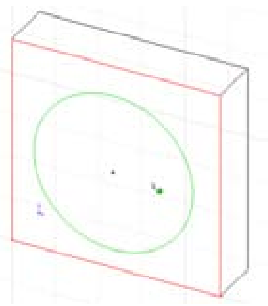
21. Klik **Hidden Lines Removed** uit de View toolbar, of **View, Display, Hidden Lines Removed**.

22. Klik **Select** van de menu bar of met het rechtermuismenu, wanneer deze nog niet is geselecteerd.

23. Klik het voorvlak aan om het te selecteren. De randen van het vlak worden groen om aan te geven dat het vlak is geselecteerd.

    Tip: *De cursor verandert naar* *om aan te tonen dat er een vlak is geselecteerd.*

24. Klik op **Sketch** uit de Sketch tab, of selecteer **Insert Sketch** uit het rechtermuismenu. Er verschijnt een raster op het geselecteerde vlak om aan te tonen dat dit het nieuwe schetsvlak is.



25. Klik op **Circle** uit de Sketch tab, of klik **Tools, Sketch Entity, Circle**.

26. Klik in de buurt van het midden van het vlak en sleep met de cursor om een cirkel te vormen.

Het bematen en extruderen van de Boss

Voeg de nodige bemating toe om de plaats en de grootte van de cirkel vast te leggen.

27. Klik **Smart Dimension** uit de Sketch Relations toolbar, of selecteer **Smart Dimension** uit het rechtermuismenu.

28. klik op de bovenste rand van het vlak, klik op het cirkelmiddelpunt en vervolgens op de positie waar je de bemating wilt plaatsen.

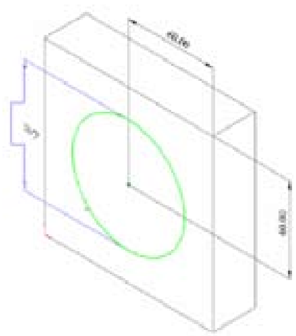
    Merk op dat de dimensioneringslijnen al even worden getoond wanneer je op de lijn en de cirkel klikt.

    Deze preview laat zien waar de aanhaallijnen (waartussen de afmeting wordt bepaald) worden vastgezet en helpt je bij het bepalen of de juiste onderdelen zijn geselecteerd. Wanneer een dimensie aan een cirkel wordt toegevoegd die de plaats van die cirkel vastlegt, is de afstandslijn gerelateerd aan het middelpunt van de cirkel.

29. Maak de afstand tussen het middelpunt van de cirkel en de lijn **60 mm**. Als je de **Input dimension value** optie aan hebt gezet, kun je eenvoudig de waarde intikken. Druk vervolgens op **Enter**.

30. Herhaal het bovenstaande proces om ook de afstand van de cirkel tot de rechterrand van het vlak te bematen. Maak deze afstand ook **60 mm**.

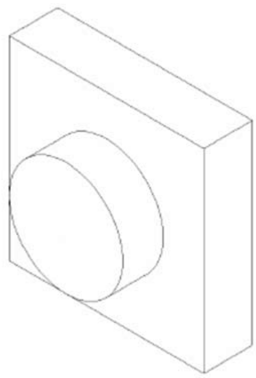
    Merk op dat de kleur van het middelpunt van de cirkel veranderd van blauw naar zwart. De positie is volledig gedefinieerd. De diameter kan op dit moment nog wel variëren.



31. Om de diameter van de cirkel te bematen blijf je de **Dimension** tool gebruiken en klikt de cirkel aan.

32. Klik zodanig dat er een diameterbemating geplaatst wordt en maak deze diameter **70 mm.** Zoals je ziet is de cirkel nu zwart gekleurd als teken dat deze nu volledig gedefinieerd is.

Klik **Extruded Boss/Base** uit de Features tab, of klik **Insert, Boss, Extrude**.



33. In de **Extrude Feature** dialog zet je **Depth** van de extrusie op **25 mm.** De andere waarden blijven ongewijzigd. Vervolgens klik je op **OK** om de boss feature te extruderen.

Merk op dat er een tweede Extrude feature in de FeatureManager design tree verschijnt.

Het maken van een gat

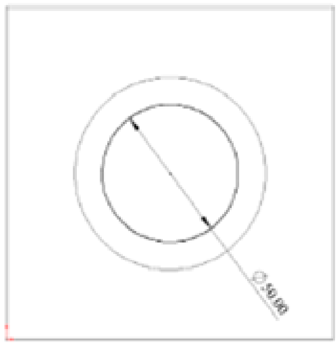
Je gaat nu een gat maken dat concentrisch ligt met de al eerder geconstrueerde boss.

34. Klik het voorvlak van de boss aan om het te selecteren.

35. In de **View Orientation** dialog uit de view toolbar klik je op **Normal To** uit de lijst met aanzichten. Het onderdeel is bijgedraaid zodat het geselecteerde modelvlak (het voorvlak) naar je toe is gericht.

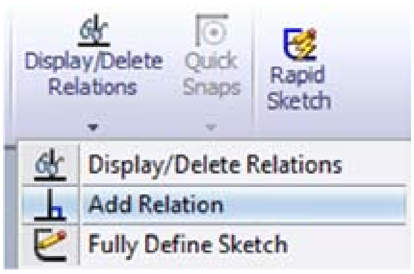
36. Open een nieuwe sketch en schets een cirkel in de buurt van het middelpunt van de boss, zoals hieronder is afgebeeld.

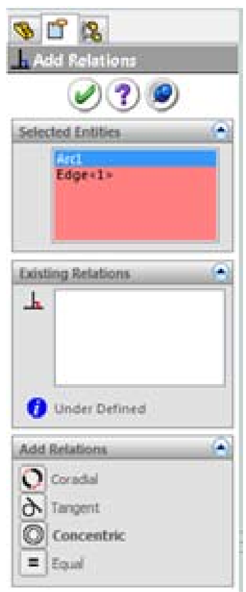
37. Voeg een bemating toe en geef de cirkel een diameter van **50 mm**.



38. Klik **Select**, of kies **Select** in het rechtermuismenu om de **Dimension** tool te deselecteren.

39. Uit de Sketch tab, klik je op het pijltje onder **Display/Delete Relations, Add Relations** of je klikt **Tools, Relations, Add...** in de Menu bar. De **Add Relation** PropertyManager verschijnt.

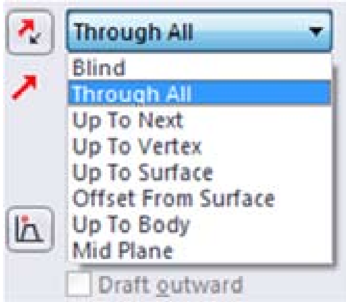




40. Selecteer de zojuist geschetste cirkel (binnenste cirkel) en de rand van de Boss (buitenste cirkel).

41. Selecteer **Concentric**, klik **OK**. Door het toevoegen van een extra relatie, zorgen we ervoor dat het middelpunt van deze cirkel op het middelpunt van de boss ligt. Zo blijft het gat altijd in het midden gepositioneerd.

42.

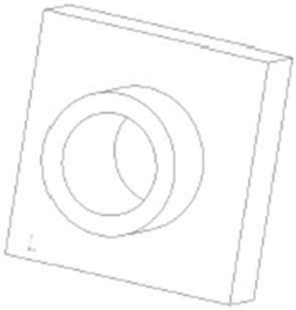


43. Klik **Extruded Cut** uit de Features tab, of klik **Insert, Cut, Extrude**.

44. Klik op het **Isometric** icoon in de standard views toolbar en bekijk de preview die van de Extruded Cut wordt gegeven.

45. In de **Cut Extrude** PropertyManager, selecteert je **Through All** als **End Condition**.

46. Bekijk de preview en klik **OK**.



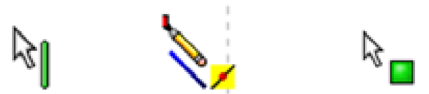
Het afronden van hoeken van onderdelen

In dit deel ga je de vier hoeken van Extrude1 afronden. Omdat de afrondingen allemaal dezelfde straal hebben (10 mm), kun je ze samen als één feature maken.

47. Klik **Hidden Lines Visible** . Dit maakt het makkelijker om de verborgen randen te selecteren.

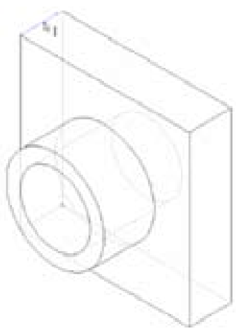
48. Klik de eerste hoek aan om deze te selecteren.

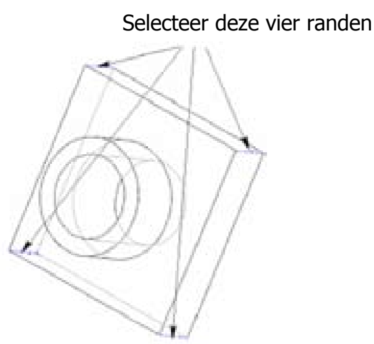
Merk op hoe de randen en vlakken oplichten wanneer de cursor er overheen gaat. Zo zie je van te voren welke onderdelen geselecteerd gaan worden. Merk ook op hoe de cursor verandert.



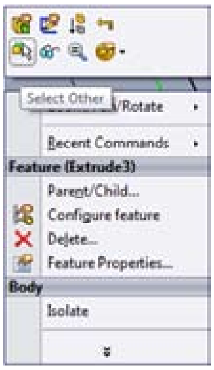
edge *midpoint* *face*

49. Houd de **Ctrl**-toets vast om de selectie uit te breiden en klik de tweede, derde en vierde hoek-rand aan. Let goed op dat de randen worden geselecteerd en niet de middelpunten.

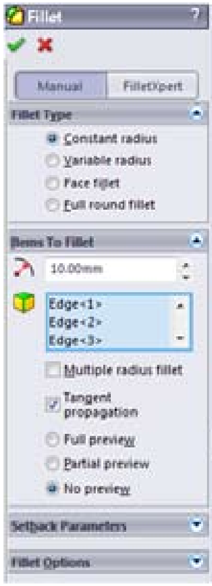


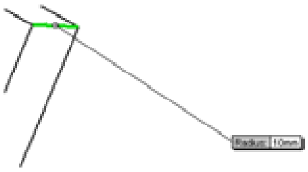


__Tip:__ Om een rand of vlak te selecteren dat achter een dichtbijgelegen vlak ligt (een verborgen rand of vlak), klikt je met de rechter-muisknop en kies __Select Other__ uit het rechtermuismenu.
Je krijgt een popup met de mogelijke opties die je wilt selecteren. Kies je Edge.



50. Klik op **Fillet** uit de Features tab, of selecteer **Insert, Features, Fillet/Round**. De Fillet Property-Manager verschijnt. Ook verschijnt er een Callout in het tekengedeelte die de Radius aangeeft.

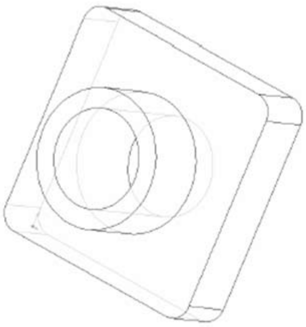




Merk op dat het **Items to Fillet** vak de vier geselecteerde randen aangeeft. Is dit niet het geval, geef dan een klik met je rechter-muisknop in het tekengedeelte, selecteer **Clear Selections** en begin opnieuw met het selecteren van de vier randen.

51. Zorg ervoor dat de **Radius** op **10 mm** staat. Laat de andere onderdelen op de standaard waarden staan.

52. Klik **OK**.

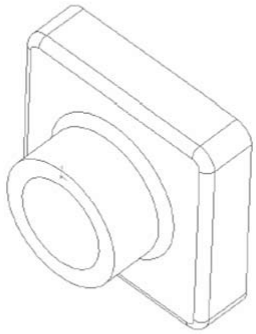


Meer afrondingen toevoegen

Je gaat nu de overige fillets (afrondingen; **Let op**, hier wordt materiaal toegevoegd) en rounds (afrondingen; **Let op**, hier wordt materiaal weggenomen) toevoegen aan het onderdeel. Je kunt randen of vlakken selecteren voor of nadat je de **Fillet Feature** dialog hebt geopend.

53. Klik of **Insert, Features, Fillet/Round**.

54. Zet het weergave type naar **Shaded with edges** .



55. Klik het voorste vlak van de base aan om het te selecteren. Zowel de binnen- als de buitenrand (van de boss) zijn gekleurd wanneer dat vlak is geselecteerd.

    Merk op dat de **Items to Fillet** lijst toont dat er één vlak is geselecteerd.

56. Verander de **Radius** naar **5 mm** en klik op **OK**. De buitenste rand van de base en de binnenste rand (van de boss) worden dan in één actie afgerond.

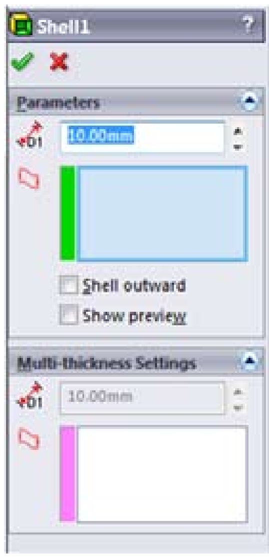
57. Klik nog een keer op .

58. Klik op het voorste vlak van de boss.

59. Verander de **Radius** naar **2 mm** en klik op **OK**.

Het uithollen van het onderdeel

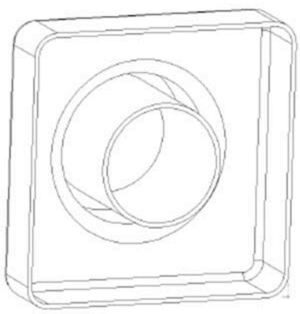
Nu wordt het onderdeel uitgehold: er wordt een shell, een dunwandig object van gemaakt. Bij het uithollen wordt materiaal verwijderd vanuit het geselecteerde oppervlak, waardoor er een dunwandig onderdeel over blijft.



60. In de **View Orientation** dialog dubbelklik je op **Back** of je klikt op het **Back** icoon in de standard views toolbar. De achterkant van het onderdeel is nu naar je toe gericht.

61. Klik **Shell** uit de Features tab, of **Insert, Features, Shell**. De **Shell** PropertyManager verschijnt.

62. Klik het achtervlak aan om het te selecteren. De shell-actie zal materiaal verwijderen vanuit dit vlak.



63. Wijzig de **Thickness** in **2 mm** en klik op **OK**.

64. Gebruik de pijltjes toetsen of houd het scroll wiel van je muis ingedrukt om het onderdeel te roteren en zo het resultaat van de shell-feature te bekijken.

65. Maak een aanzicht met bijbehorende naam:
    - Klik op de **Spatiebalk**, en vervolgens in het **View Orientation** dialog op de **New View** button.
    - Typ een naam als **Shell Back** in het daarvoor bestemde **Named View** dialog in en klik op **OK.**

Opslaan van een onderdeel

66. Klik **Save** uit de Standard toolbar, of klik **File, Save**. Omdat het onderdeel nog geen naam heeft, verschijnt de **Save as** dialog.

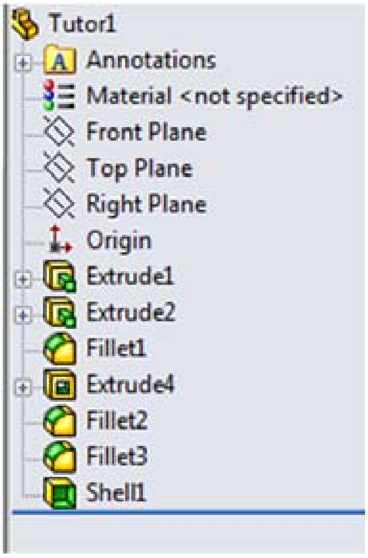
67. Vul bij **Save in** de plaats in waar het bestand moet komen te staan.

68. Vul bij **File name** de naam **'TUTOR1'** in. SolidWorks voegt de extensie .SLDPRT toe aan de filenaam.

__Opmerking:__ SolidWorks maakt bij het saven van File-namen geen onderscheid tussen hoofdletters en kleine letters, dus __TUTOR1.SLDPRT, Tutor1.SLDPRT en tutor1.SLDPRT__ zijn allemaal hetzelfde onderdeel.

Het wijzigen van een bemating

Nu wordt uitgelegd hoe bematingen kunnen worden aangepast.

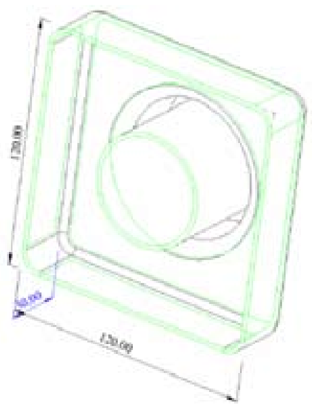


69. Bestudeer de FeatureManager design tree. Deze toont de features van het onderdeel in de volgorde waarin ze gemaakt.

70. Dubbelklik op de eerste **Extrude** (van **stap 20**) in de FeatureManager design tree.

Merk twee dingen op (klik eventueel op **Zoom To Fit** om alle features te tonen):

- In de FeatureManager design tree is de **Extrude** feature uitgebreid om de sketch te tonen waaruit hij is gemaakt.
- In het tekenvenster licht de geëxtrudeerde feature op en zijn de afmetingen weer gegeven.



71. Dubbelklik op de afmeting voor de diepte van de extrusie (30.00). De **Modify** dialog verschijnt.

72. Verander de waarde van **30 mm** naar **50 mm**.

73. Klik op **Rebuild** in de **Modify** dialog.

SolidWorks bouwt het onderdeel nu weer opnieuw op, waarbij gebruik wordt gemaakt van de nieuwe afmetingen.

__Opmerking:__ je kunt meerdere dimensies in één keer wijzigen en dan pas het onderdeel met alle wijzigingen opnieuw opbouwen. Verander dan elke waarde waarna je op de __Check__ knop drukt in de __Modify__ dialog. Wanneer je tevreden bent met alle wijzigingen klik je op __Rebuild__ uit de Menubar, of selecteer je __Edit, Rebuild__ in de Menu bar.

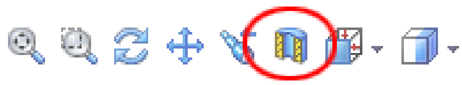
74. Klik op **OK** om het **Modify** dialog te sluiten, en klik op **Save** om het onderdeel te bewaren.

Weergave van een doorsnede

Je kunt op elk moment een 3-dimensionale doorsnede van een onderdeel construeren. Om het doorsnedevlak te specificeren kun je onderdeelvlakken of tekenvlakken (Front-,Top- of RightPlane) gebruiken. In dit voorbeeld maak je gebruik van Right Plane om de doorsnede te maken.

75. Ga over naar het **Isometric** aanzicht.

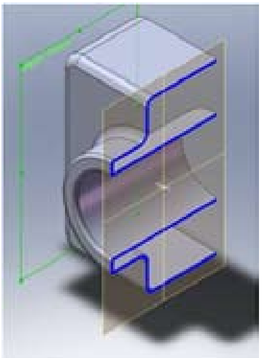
76. Klik op het **Right** plane in de FeatureManager design tree.



77. Klik op **Section View**  in de view toolbar. De Section View dialog verschijnt.

78. Stel de **Offset Distance**  in op **60 mm**. Dit is de offset-afstand vanaf het geselecteerde oppervlak naar het doorsnedevlak.

79. Klik **OK**.



De doorsnede van dit onderdeel is weergegeven. Alleen de weergave van dit onderdeel is doorgesneden, niet het daadwerkelijke model. De doorsnede blijft gehandhaafd ook wanneer een aanzicht, weergave (shaded, wireframe, etc.) wordt gewijzigd of wanneer je zoomt.

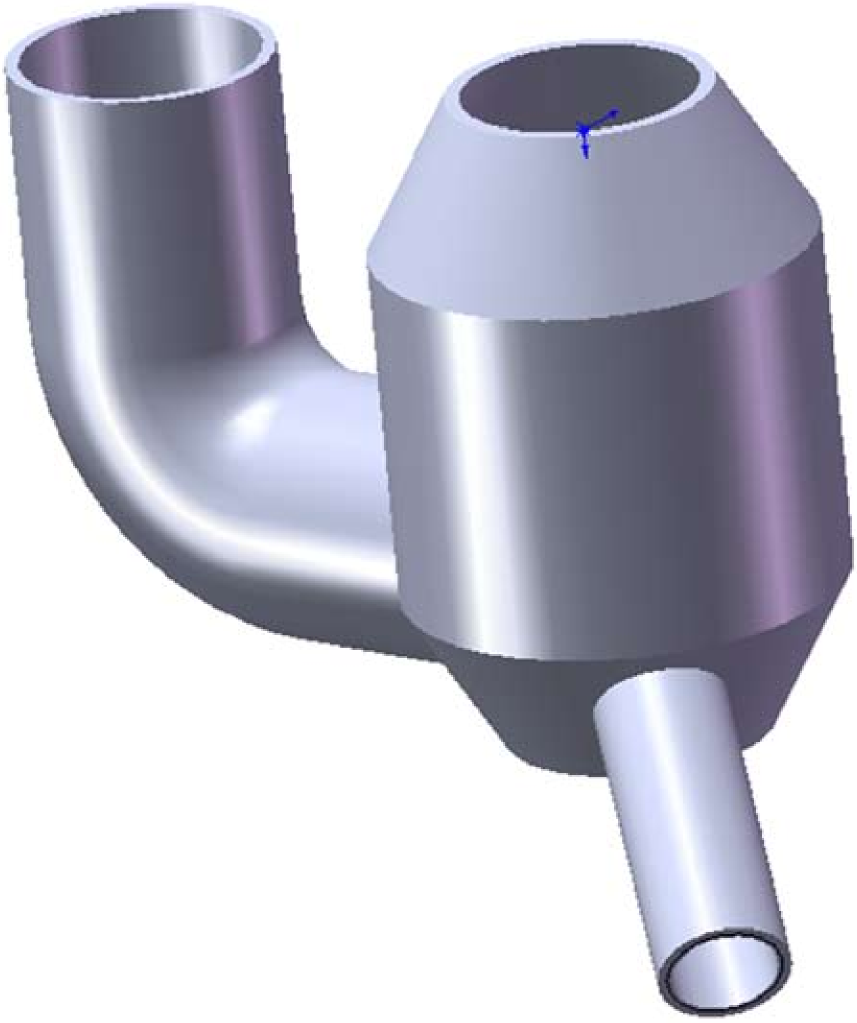
80. Om weer het hele onderdeel afgebeeld te krijgen, klik je weer op **Section View**.

81. Save het part in je */work directory/*, met filename **'TUTOR1'**, als je dit nog niet hebt gedaan.

Vanaf hier is het dictaat in het Engels.

Sweep and Revolve Features

In this section the Sweep and Revolve features are treated. It will be explained how to create the model below.



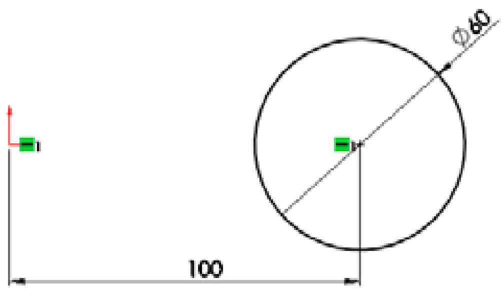
Sketching the Sweep Section and Path

A sweep is a base, boss or cut created by moving a section along a path. First, you sketch the sweep section on the Front reference plane.

1. Open a new part.

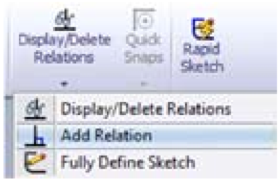
2. Click the **Sketch** tool to open a sketch on the **Front** plane.

3. Click **Circle** , or **Tools, Sketch Entity, Circle**, and sketch a circle with a **Diameter** of **60 mm**, with its **centerpoint 100 mm** from the origin of the **Front** plane.



4. Dimension the sketch.

5. On the Sketch tab, click the arrow under **Display/Delete Relations, Add Relations** or click **Tools, Relations, Add…** on the Menu bar. The **Add Relation** PropertyManager appears.



6. Select the origin and the midpoint of the circle and add the relation **Horizontal**. The circle turns black, which means that the sketch is fully defined.

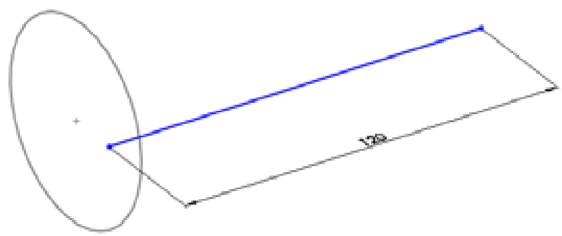
7. Click **Exit Sketch** to close the sketch.

   Next, sketch the sweep path. The path can be an open curve, or a closed, non-intersecting curve. Neither the path nor the resulting sweep may self-intersect. The end point of the path must lie on the plane of the section.

8. Select the **Top** plane in the FeatureManager design tree and open a new sketch.

9. Click the **Isometric** icon in the Standard Views toolbar, and zoom out until you can see the origin.

10. Click **Line** , or **Tools, Sketch Entity, Line**, and sketch an **120 mm** vertical line anywhere perpendicular to the circle, as is shown below. Define the **120 mm** length by adding a **Smart Dimension**.



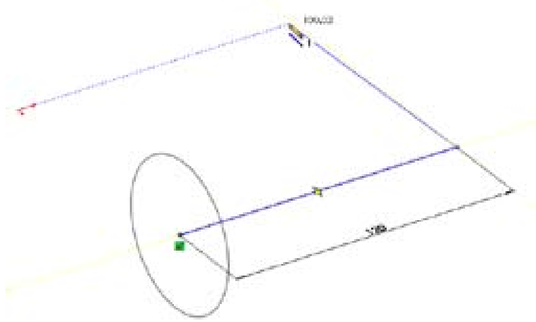
11. Deselect the Line tool by clicking **Select** . The line is automatically deselected, so select it again. In the propertymanager to the left you can see the relations of the line. we want the line to be exactly vertical, so add a **Vertical** relation if it doesn't exist already and click **OK** .

    Tip: *Watch the cursor for feedback about the length of the line, and for automatic relations. The cursor has a yellow block with a vertical line attached to it when the line is vertical. This means that SolidWorks will automatically add an relation. As you sketch, inferencing lines and cursors help you align the cursor with existing sketch entities and model geometry. You can find more information about* **Inferencing Pointers and Lines** *in the SolidWorks Online User's Guide.*

12. Select the midpoint of the circle and the endpoint of the line nearest to the circle using the ctrl key. Click **Coincident** in the **Add Relations** PropertyManager, then click **OK**. This relation insures that the end of the path lies exactly on the plane of the circle.

13. Sketch a **100 mm** horizontal line attached to the end of the first line, opposite of the circle. Add the dimension.

    Notice the blue dashed inferencing line when the cursor is aligned with the origin.



14. We want the second line to be perpendicular to the first. So deselect the Line tool, and select both lines. In the propertymanager, add a **Perpendicular** relation.

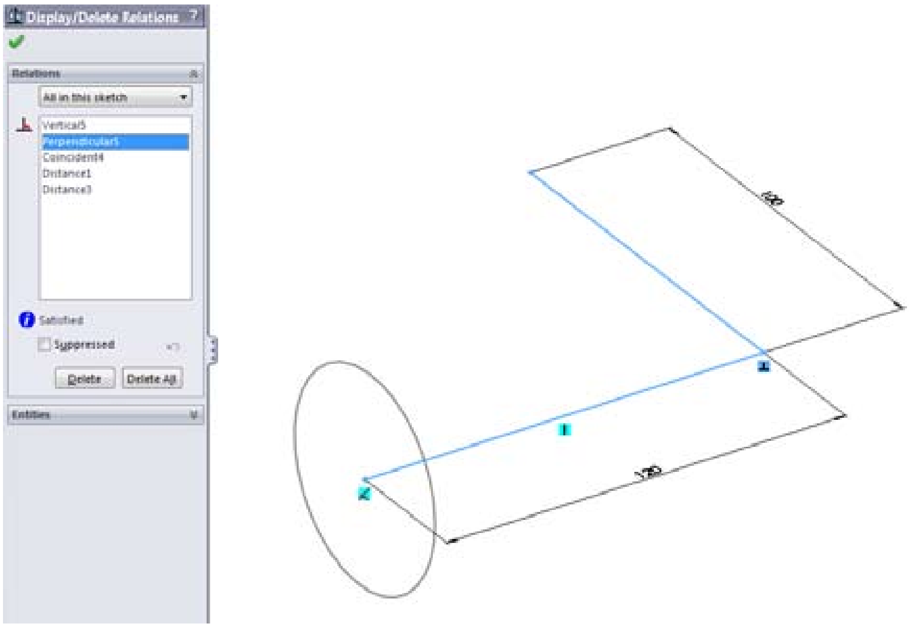
Before continuing, we will use **Display/Delete Sketch Relations** to check if the proper relations are added correctly.

15. Click **Display/Delete Relations** or **Tools, Relations, Display/Delete**.

The **Sketch Relations** PropertyManager appears. It provides access to a list of all the relations in the current sketch, including those that are added automatically as you sketch (such as a **Horizontal** relation added by inference), and those that you add manually.

16. In the **Relations** box, be sure that **All in this sketch** is selected in the **Filter** pull down list.

17. Click a relation in the **Relations** box. As you do, the entities involved are highlighted. Make sure that the just created relations (**Vertical**, **Coincident** and **Perpendicular**) are present and are related to the right entities. The dimensions of the lines are also in the list. Click **OK** to exit the Relations PropertyManager.



You can also display the relations for one entity at a time. In the **Relations** *box, choose* **Selected Entities** *from the* **Filter** *pull down list and then select a sketch entity.*

Next, we will add a curve to the corner of the lines, using **Sketch Fillet**.

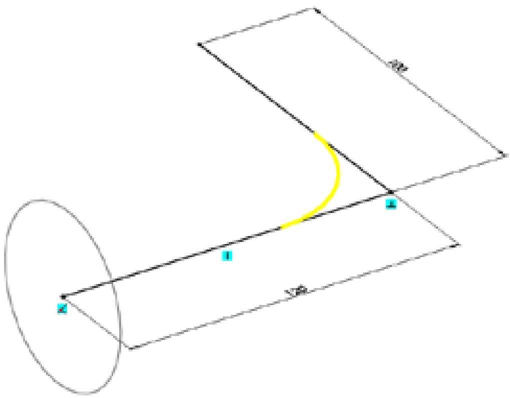
18. Click **Sketch Fillet** . The **Sketch Fillet** PropertyManager appears.

19. Set the Fillet Radius to **40 mm**.

20. Select the corner of the two lines. A Yellow preview appears.

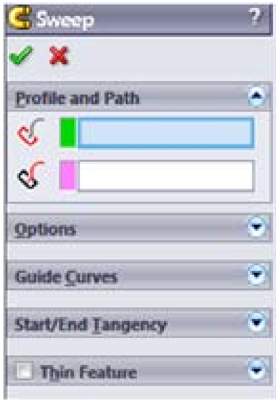
21. Click **OK**. An arc is added in the corner between the lines. Notice that on both ends a tangent relation is automatically added between the arc and the line.

22. Click **OK** again to exit the **Sketch Fillet** PropertyManager.



Creating the Sweep

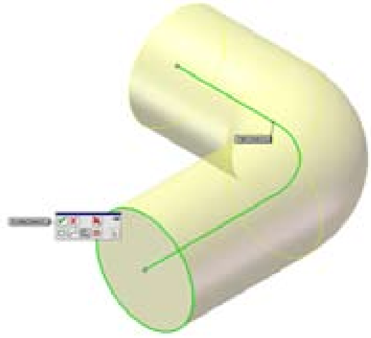
Now you can combine the two sketches to create the sweep.



23. Click on the Feature tab, or click **Insert, Base, Sweep**. The **Sweep** PropertyManager appears.

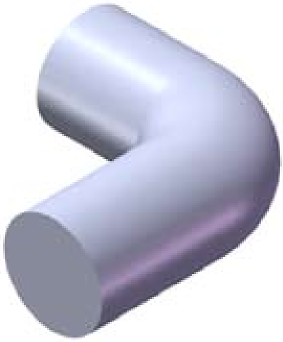
24. Click the **Sweep Profile** box, then select the first created **Sketch** in the FeatureManager design tree by clicking on the Sweep PropertyManager title (or click the circular section in the graphics area).

25. The **Path** box is automatically activated. Click the second **Sketch** in the FeatureManager design tree (or click the path in the graphics area).

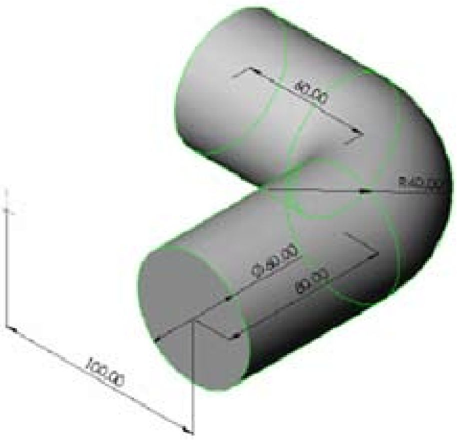


26. Make sure the **Orientation/Twist Type** is set to **Follow Path**.

27. Click **OK** to create the sweep.



__Tip:__ You can toggle the display of dimensions by right-clicking the __Annotations__ folder in the FeatureManager design tree, and checking __Show Feature Dimensions__. To temporarily display the dimensions, double-click the __Sweep__ in the FeatureManager design tree. Click anywhere in the window to remove the dimension display.



28. Save the part in */work directory/* as **Verdeelstuk.SLDPRT**.

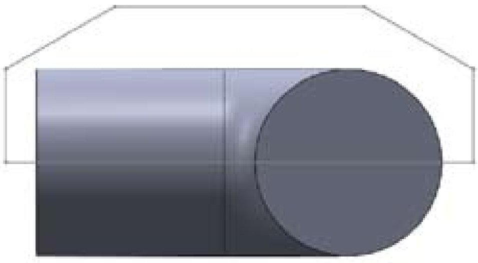
Sketching a Revolve Profile

Much like a sweep, a revolve uses a moving profile. The difference is that instead of a path, you sketch a centerline around which the profile revolves. In addition to the dimensions, you will use several relations to control the shape, position and behavior of the profile.

29. Click on the face of the longer end of the sweep, and open a new sketch.

30. In the Standard Views toolbar click **Normal to**.

Now you will create the sketch shown below. It consists of a horizontal line through the origin (which will be the center of the revolve), plus a number of connected lines, forming a closed contour.



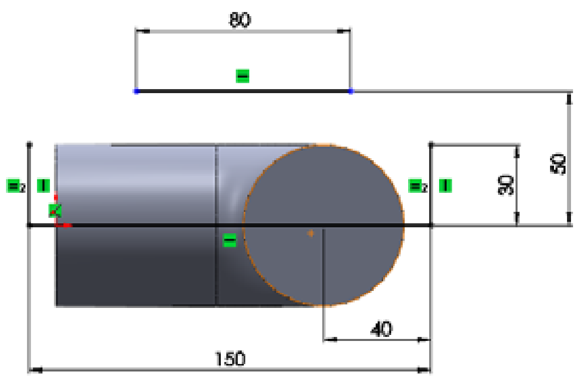
31. Sketch the horizontal line through the origin (note the on-axis cursor shape).

32. Sketch a vertical line at the left and right side of the profile. Add an **Equal** relation between the two vertical lines.

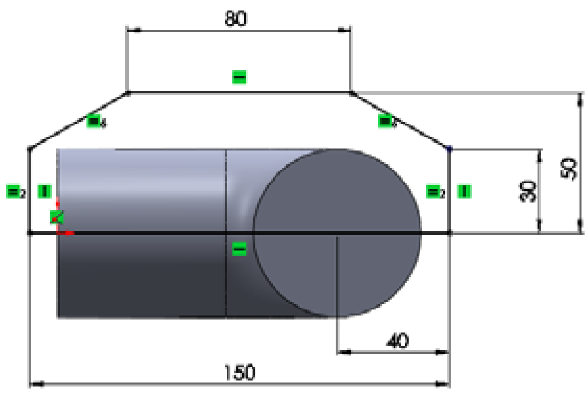
__Tip:__ By control-selecting both lines, the Add Relation dialog will appear automatically.

33. Sketch a top horizontal line anywhere above the part.

34. Dimension the lines as shown to the right. Do not be concerned about the horizontal location of the top line yet. The horizontal 40 mm dimension starts in the center of the circle. To access this center, simply click the edge of the circle. By default a dimension between a circle and another sketch entity is a dimension between the circle's center and the other entity.

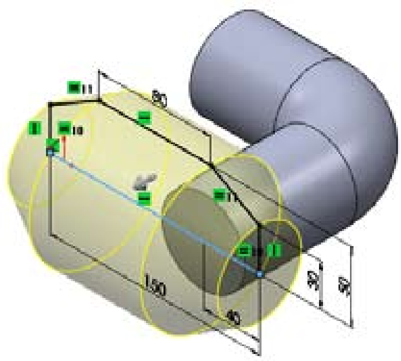


35. Sketch the two diagonal lines, and add an **Equal** relation between them. Notice the effect on the angles and on the top line (from **step 28**).



Creating the Revolve Feature

After creating the sketch, we can revolve it around a centerline, in our case the bottom line, to create a round solid feature.



36. Select the Bottom horizontal line and click **Revolved Boss/Base** on the Features tab, or **Insert, Boss/Base, Revolve**... The **Revolve** PropertyManager appears.

37. Leave the default value of **Angle** at **360°**. Do not check the **Thin Feature** checkbox (this would create a hollow compartment with a predefined wall thickness).

38. To observe the preview better, change the **View Orientation** to **Isometric** .

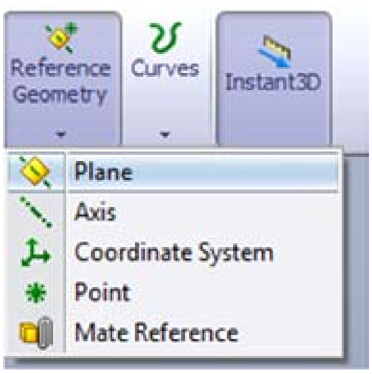
39. Click **OK** to create the revolve.

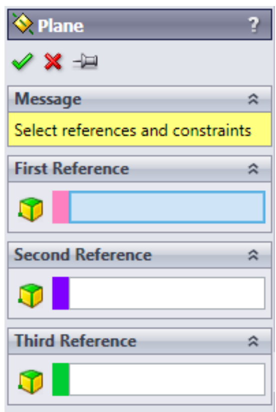
Creating New Reference Planes

Apart from the basic three Planes, you can also create your own **Reference Planes**.

40. First, click **View, Temporary Axes** to display the axes running through all circular features of the part (the blue dashed lines).

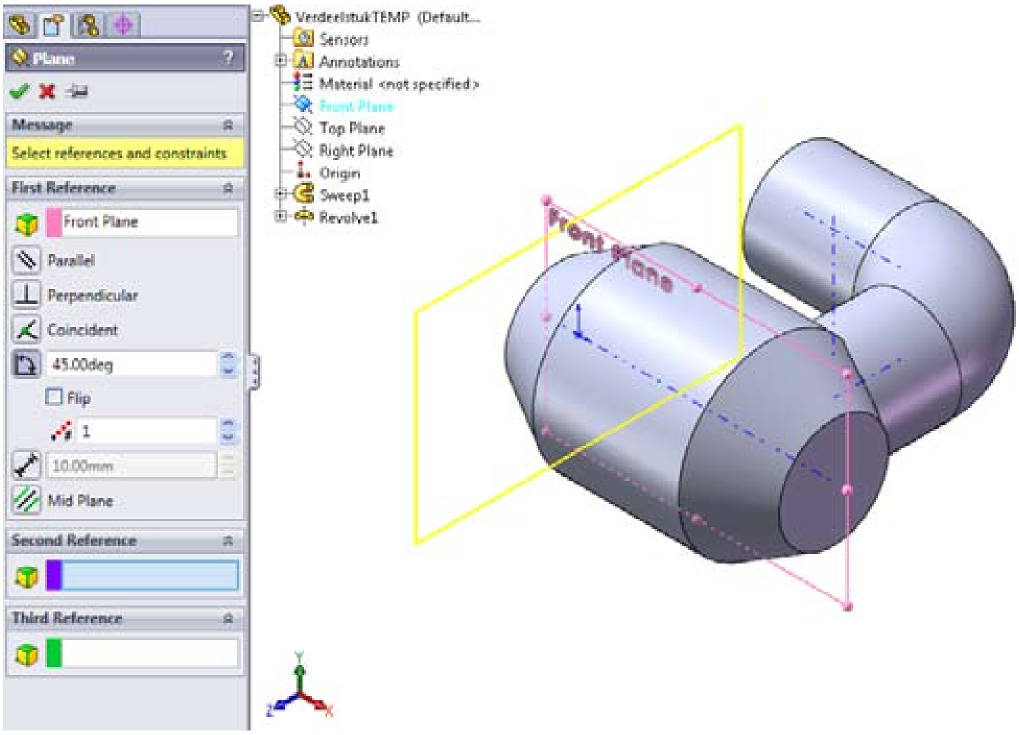
41. Click **Reference Geometry** , **Plane** , or click **Insert, Reference Geometry, Plane**. The **Plane** PropertyManager appears.



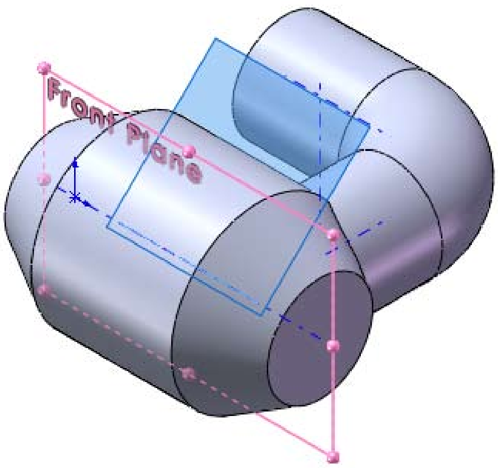


42. In the **First Reference** box, Select the **Front** plane from the FeatureManager design tree, which you can bring to the front by clicking the title bar of the **Plane** PropertyManager, or by clicking the + next to **Sweep** at the top left corner of the drawing window. the available options with this reference appears.

    Click **At Angle** . The Message box turns yellow, indicating that it needs more reference material.



43. Select the blue dashed **axis** that runs through the revolved feature as **Second Reference**. Solidworks can now put the new plane at an angle in relation to the axis. The plane shifts to the right position and the Message box turns green stating that it is **Fully Defined**.

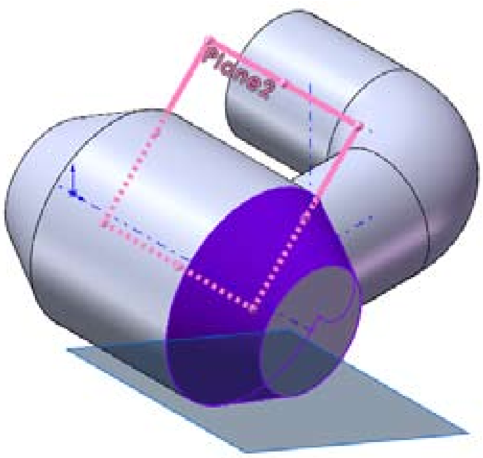


44. Specify an **Angle** of **45°**, and click **Reverse** if necessary to tilt the plane as shown here.

    Tip: *You can see the plane rotate as you change the value in the Angle spinbox or as you select the Reverse checkbox.*

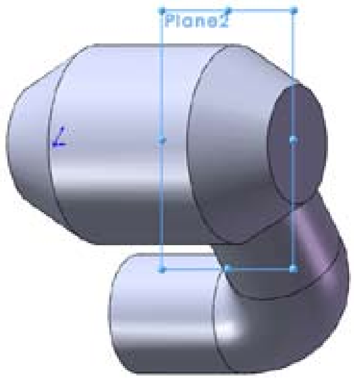
45. Make sure that the **pushpin** -> is pressed (we want to make another plane so the PropertyManager should remain visibly after clicking OK) and click **OK** to create the plane. the first plane you just created is automatically selected. Select **Perpendicular** as the reference type.

46. We will now create a plane on the lower angled face of the revolve. Select the face as **Second Reference**. SolidWorks selects **Tangent** as reference type.



47. The preview should look like shown to the right. If not, check **Flip** to obtain the shown situation.

48. Click **OK** to create the plane.



The planes are shown in the FeatureManager design tree to the left and will highlight if you select them.

49. Select the new plane from the FeatureManager design tree and change the **View Orientation** to **Normal To** . The part turns so that the new plane is facing you.

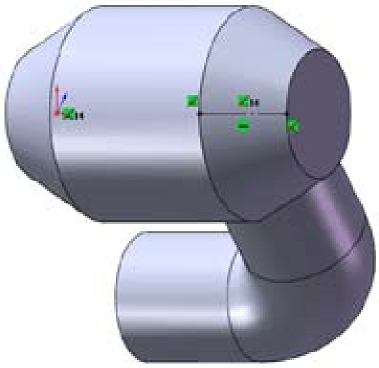
Tip: To hide the axis lines, click **View, Temporary Axes** *again.*

Adding a Feature on a New Plane

50. Open a new sketch on the selected **Plane**.

51. Select the **Line Tool**. before drawing any lines, check **For Construction** in the **Options** box.

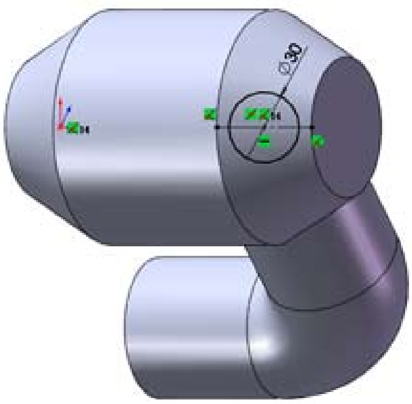
A **Construction Line** *will not be used by a feature. It is only used for constructing the sketch (hence the name).*



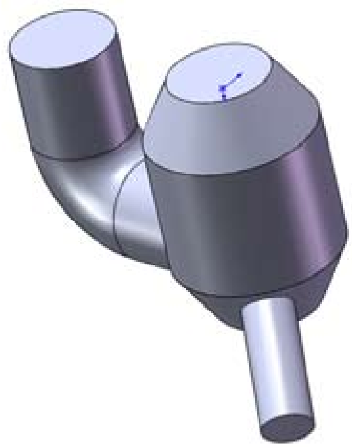
52. Draw a line between the two edges of the angled face. Give it a **Horizontal** relation. Also add a **Coincident** relation between the line and the **Origin**.

53. Sketch a **circle** anywhere, and dimension it **30 mm** in diameter.

54. Select the **center** of the circle and the **Construction Line**. Add a **Midpoint** relation. The circle is now exactly in the middle of the face.



55. Click or **Insert, Boss, Extrude**.



56. In the **Direction 1** group box, set **End Condition** to **Blind** and set the **Depth** to **75 mm**.

57. Expand the **Direction 2** group box, and check the **Direction 2** option. Set the **End Condition** to **Blind** and set the **Depth** to **5 mm**.

58. Click **OK**, then rotate the part to see the new extrusion.

59. **Save** the part.

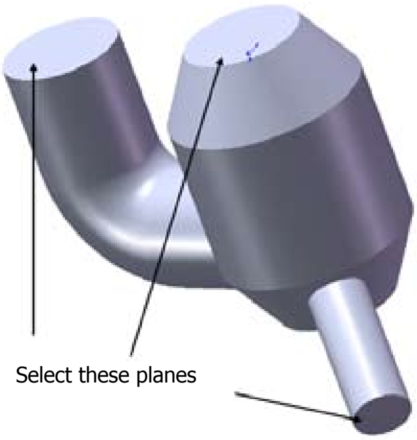
Creating a Multi-face Shell

60. Rotate the part until the face at each of the pipe ends is in view.

61. Click **Shell** or **Insert, Features, Shell**.

62. In the **Shell** PropertyManager, set the **Thickness** to **3 mm**.

63. Click the three faces indicated to the right. Notice the three faces in the **Faces to Remove** list.



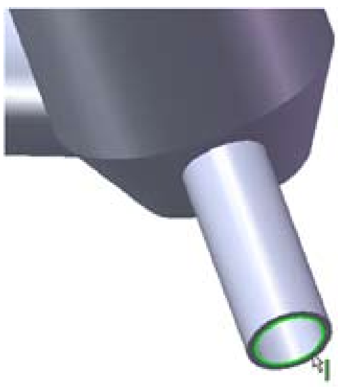
Note: *If you click the wrong face accidentally, click it again to deselect it. You could also select its name in the* ***Faces to Remove*** *box and press* ***Delete****. Alternatively, you could right-click in the graphics area, and select* ***Clear Selections****.*

64. Click **OK**.

SolidWorks creates a multi-faced shell, removing material from each of the selected faces, leaving 3 mm walls.

Adding screw thread

65. Select the inner edge of the hole in the tube indicated in the picture below.

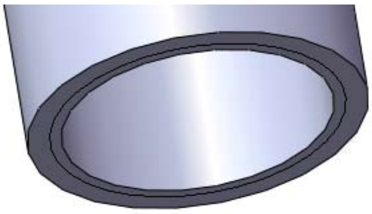


66. Click **Insert, Annotations, Cosmetic Thread**. The **Cosmetic Thread** dialog appears.

67. Set the **Major Diameter**  to **26.00 mm.**

68. Set the **End Condition** to **Blind**, and the **depth**  to **30.00 mm**.

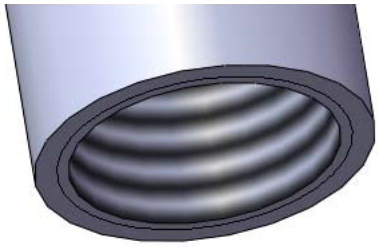
69. Click **OK**.



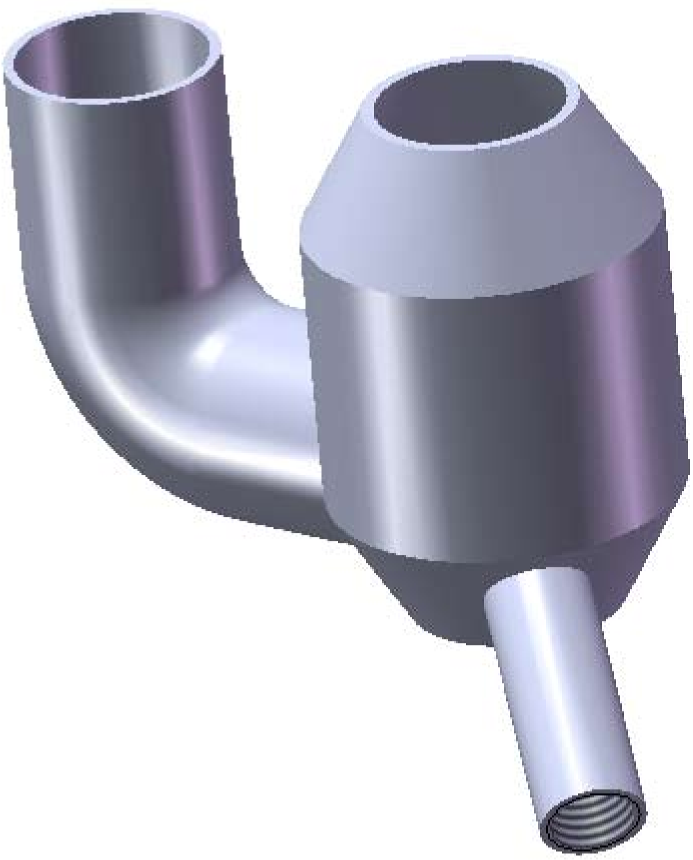
As you can see, SolidWorks doesn't show the whole thread, but only a circle on the outside face. SolidWorks does this to improve its performance. You can change this setting in the **Options** menu.

70. Click **Tools, Options...**

71. Go to **Document Properties, Detailing** and check the **Shaded Cosmetic Thread** box. Click **OK**. the thread should now look like the example to the right.

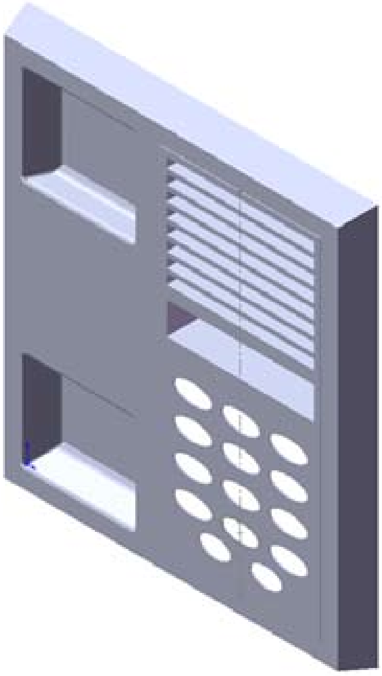


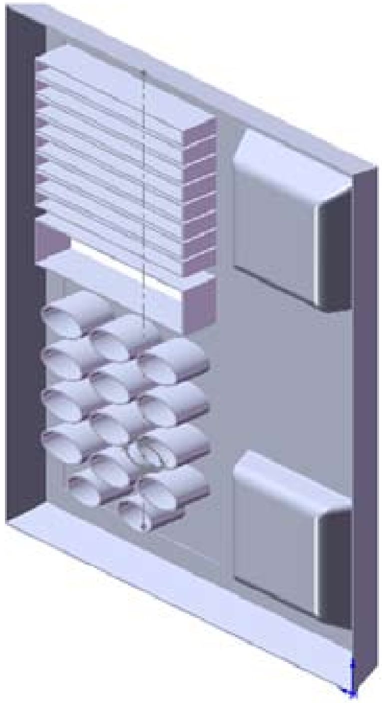
72. The part is now finished. Save your work and close the part.



Creating a Linear Pattern

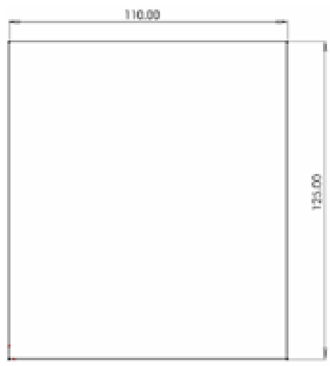
In this part it will be explained how a linear pattern can be created. You will learn to draw the model shown below.





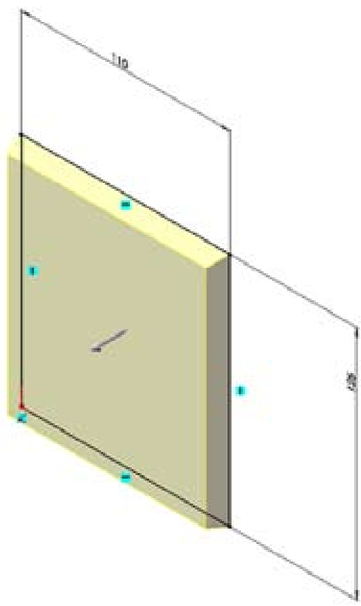
Creating the Base

First you create a base, with a raised panel for the speaker and buttons.



1. Open a new part with your own template.

2. Open a sketch, and sketch a **110 mm x 125mm** rectangle, with the lower left corner at the origin.



3. Click or **Insert, Base, Extrude**.

4. Set **End Condition** to **Blind**, and specify a **Depth** D of **10 mm**.

5. Click **Draft While Extruding**, set the **Angle** to **15°**, and deselect **Draft Outward**.

6. Make sure Thin Feature is deselected.

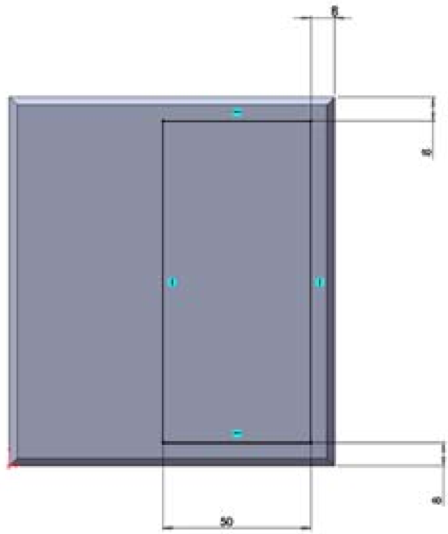
7. Click **OK**.

8. Click the front face of the base and open a sketch.

9. Sketch a long rectangle, and dimension as shown below.

   Tip: *If you dimension the distance between the rectangle and the edges of the base (instead of dimensioning the size of the rectangle itself), the feature's size will change if the size of the base changes.*

10. Click or **Insert, Boss, Extrude**. Extrude the sketch as a solid feature, to a **depth** of **0.25 mm**.



Creating a Linear Pattern of Slots

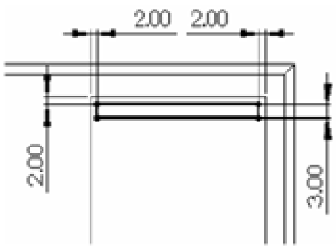
Now cut a slot, then create a vertical linear pattern of slots for the speaker.

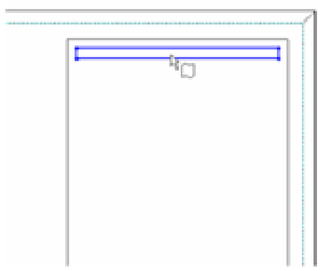
11. Click the face of the raised panel and open a new sketch.

12. Sketch and dimension a rectangle as shown below.

13. Click or I**nsert, Cut, Extrude**. Set the **End Condition** to **Through All**, and click **OK**.

    This is the feature that you will repeat to create the pattern.



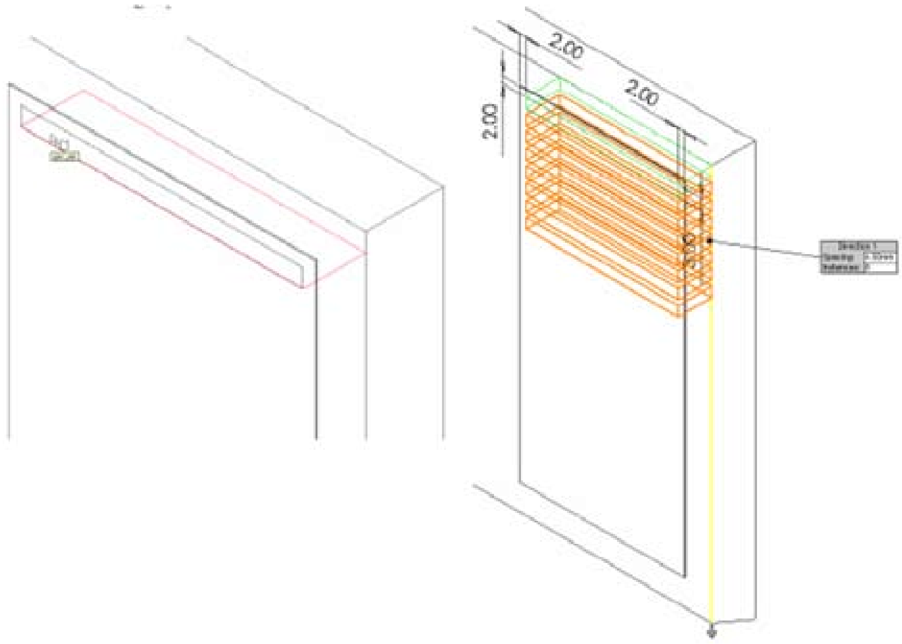


14. You can rename features from their generic names to something more meaningful:
    - Click twice on the **Cut-Extrude** of step 11-13 in the FeatureManager design tree (do not double-click; you must pause slightly between clicks).
    - When the **Cut-Extrude** appears highlighted in a box, enter the new name, **spkr_slot**, and press **Enter**.

15. Click , or **Insert, Pattern/Mirror, Linear Pattern**. The **Linear Pattern** PropertyManager appears.

    When the PropertyManager of a feature is shown, it covers the FeatureManager design tree. Thanks to the **Flyout FeatureManager design tree**, however, you can display both the PropertyManager and the FeatureManager design tree at the same time. Just click the PropertyManager title or the FeatureManager design tree tab. To hide the FeatureManager design tree again, click somewhere in the graphics area.

16. Click the **spkr_slot** feature in the Flyout FeatureManager design tree, or click one of its faces in the graphics area.



17. To set the **First Direction**, click any **vertical** edge on the part. An arrow appears to indicate the direction the pattern will take. If the arrow is not pointing down, click **Reverse Direction** . A Callout is attached to the line you selected for the direction. This callout displays the main properties of the pattern. As you proceed, a preview of the pattern is displayed.

18. Set **Spacing** to **4.5 mm**. This value is the distance from a point on one instance of the patterned feature to the corresponding point on the next instance.

19. Set **Number of Instances** to **8**. This value includes the original cut-extrude feature.

20. Click **OK** to accept the pattern.

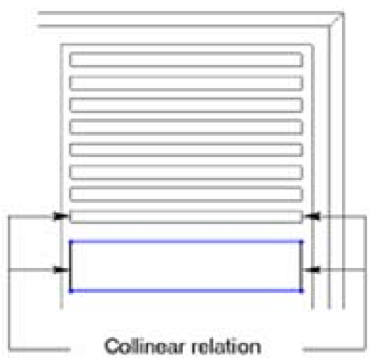
21. Rename the pattern feature **slot_pattern**, as described in Step 14.

22. Save the part in */work directory/* as **telephone.sldprt**.

Adding a Cut Feature

23. Press the spacebar to use the **Orientation** dialog and double-click **Front**.

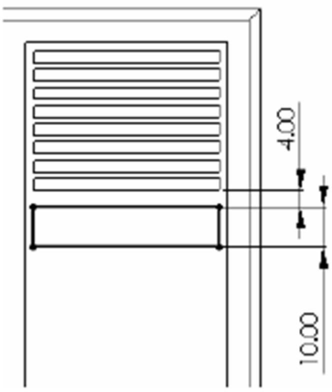
24. Click the face of the raised panel, and open a new sketch.



25. Sketch a rectangle below the slot pattern, then choose **Select** from the right-mouse menu to deselect the rectangle.

26. Relate the width of this rectangle to the width of the slots:
    - Click or **Tools, Relations, Add**.
    - Click a vertical side of the rectangle and a vertical edge of one of the slots.
    - Click **Collinear** , and click **OK**.

Repeat for the other side of the rectangle.

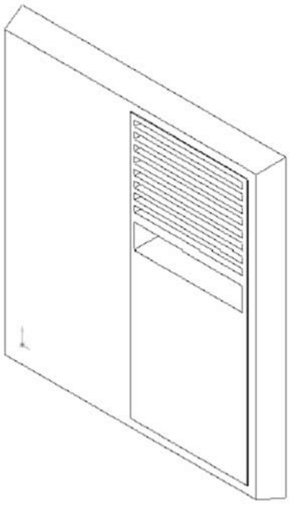


27. Add dimensions as shown to the right.

28. Click or **Insert, Cut, Extrude**.

29. Set the **End Condition** to **Through All** and click **OK**.

30. Rename the feature **msg_window**.



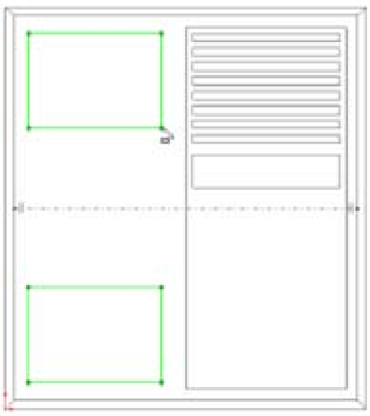
Creating the Handset Cradle

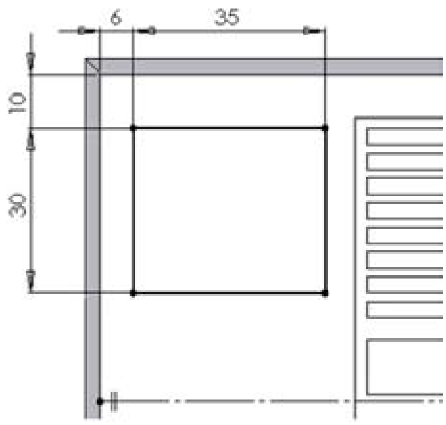
Now cut and shape the indentations that hold the telephone handset.

31. Open a sketch on the large front face of the base (not the raised panel section).

32. Sketch a horizontal centerline across the midpoints of the face, note the shape of the cursor.

33. Sketch a rectangle.





34. With the rectangle still selected, click or **Tools, Sketch Tools, Mirror**.

35. When you select the centerline, a mirrored copy is sketched on the opposite side of the centerline.

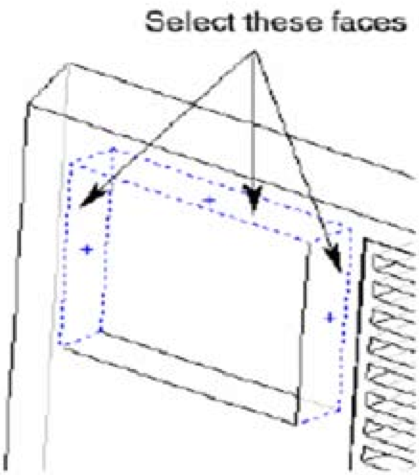
36. Dimension as shown.

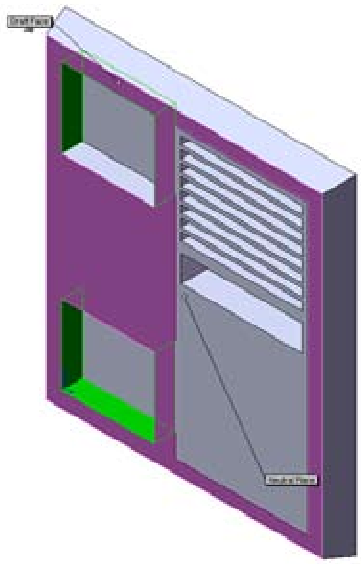
37. Cut to a **depth** of **8 mm**.

Three sides of each indentation need a draft angle

38. Click **Draft** or **Insert, Features, Draft**.
  - Select Manual, if not already selected at the top.
  - Make sure the Type of Draft is set to Neutral Plane.
  - Set the Draft Angle to **25°**.
  - Click the front face of the part. Its name is listed in the Neutral Plane box. In the graphics area the face is colored purple (corresponding to the colored bar in the Neutral Plane box of the PropertyManager) and a callout is attached to easily identify the face as the neutral plane.

39. Click the **Faces to Draft** box, then click the three sides of each indentation indicated (total of six faces). In the graphics area the faces are colored green (corresponding to the colored bar in the Faces to Draft box of the PropertyManager) and a callout is attached to the first face to easily identify the faces.





Tip: *To make face selection easier:*
- *Change to Hidden Lines Removed display mode.*
- *Use the right-mouse menu Select Other function*
- *Set the Selection Filter to Faces (If the Selection Filter is not displayed, click View, Toolbars, Selection Filter).*

40. Click **OK**.

41. Save the part.

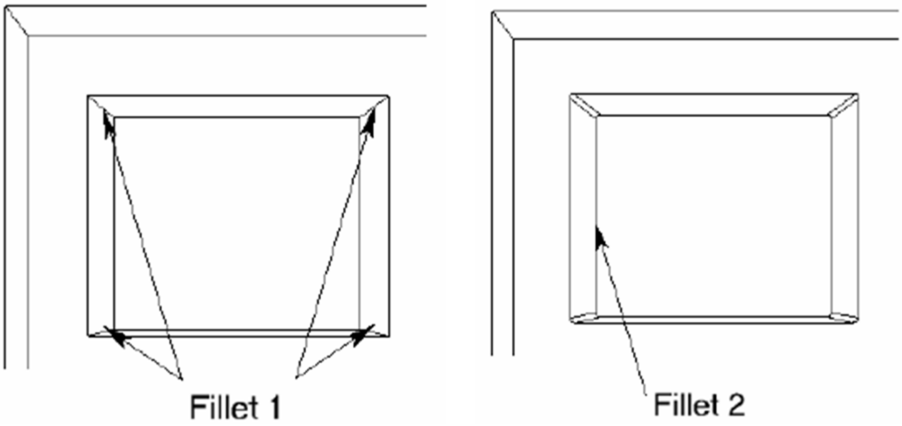
Finishing the Handset Cradle

Fillets complete the cradle shape.

42. Select **Selection Filters** under **View, Toolbars** to turn on the selection filters toolbar.

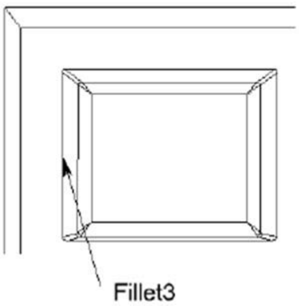
43. Set the **Selection Filter** to **Edges** by clicking in the Selection Filter toolbar.

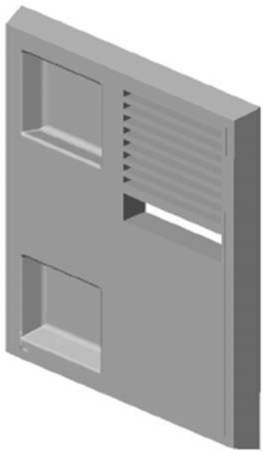
44. Click **Fillet**, and add a **0.8 mm** fillet (**Fillet1**) on the four vertical edges of both indentations. If necessary, you can tilt the model using the arrow keys while you are selecting the edges.



45. Add a **2.5 mm** fillet (**Fillet2**) around the bottom edges of both indentations. You only need to select one edge segment of each indentation; all tangent edges are filleted by default.

46. Add a **1.2 mm** fillet (**Fillet3**) at the top edge of one indentation.





47. You can copy a fillet feature from one edge or face to another using drag-and-drop. Press the **Ctrl** key, then drag the **Fillet3** feature from the FeatureManager design tree and drop it on the top edge of the second indentation.

48. Deselect the Selection Filter **Edges**.

Adding Holes for the Buttons

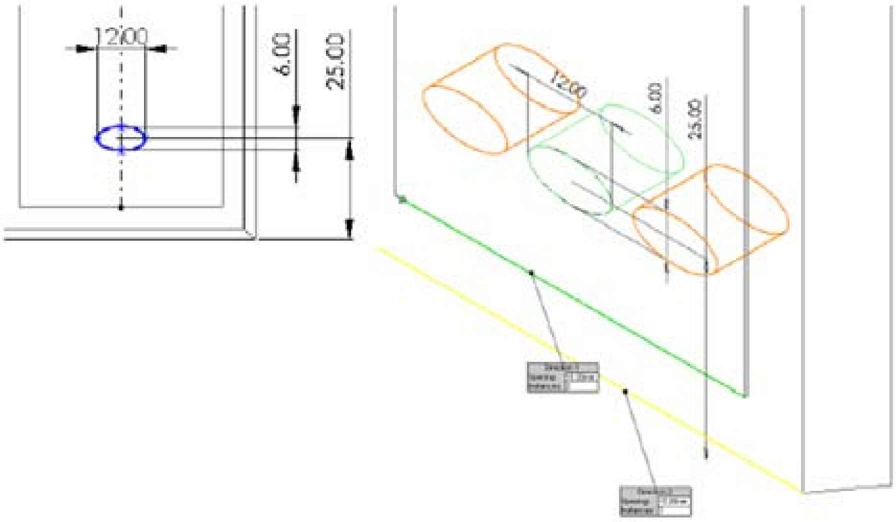
The elliptical button holes are centered horizontally on the raised panel.

49. Open a sketch on the raised panel face. Sketch a vertical centerline across the midpoint of the panel.

50. Click or **Tools, Sketch Entities, Ellipse**. To sketch the ellipse, drag horizontally from the centerpoint of the ellipse (on the vertical centerline) to set the size of one axis, release the mouse, then drag vertically to set the other axis. Watch for horizontal and vertical interferencing lines as you drag; otherwise, the ellipse may be slanted.

51. Dimension as shown below. Fully define the sketch by selecting the two points to the left and right of the center point and the center point itself and adding a **Horizontal** relationship in the PropertyManager.

52. Extrude the sketch as a through all cut. Rename this feature **button1**.



53. With this cut feature still selected, click **Linear Pattern** .
   - To set **Direction 1**, click any horizontal edge.
   - Set **Spacing** D to **15 mm** and **Number of Instances** to **2**, and observe the preview.
   - Click in the **Direction 2** box, then click a horizontal edge again.
   - Use the same values for **Spacing** and **Number of Instances**, and be sure that the **Reverse Direction** button is pressed.
   - Click **OK**.

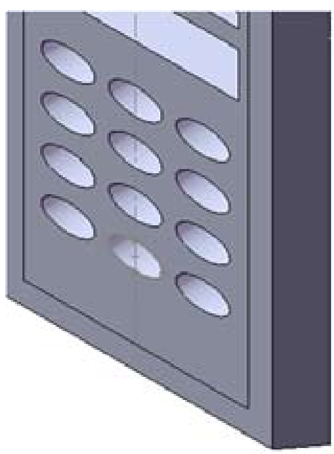
54. Rename the pattern feature **btn_row1**.

Creating a Pattern of a Pattern

You can make rows of holes by inserting a pattern of the pattern you just made.

55. click **Linear Pattern** .

56. Click the horizontal pattern (**btn_row1**) in the FeatureManager design tree (remember that you can display the Feature-Manager design tree next to the PropertyManager by clicking the name of the PropertyManager). Its name appears in the **Features to Pattern** list box.



57. Click a vertical edge or dimension to set **Direction 1**. If the arrow does not point up, click **Reverse Direction** .

58. Set **Spacing** to **10 mm**, and **Number of Instances** to **4**.

59. Click **OK**.

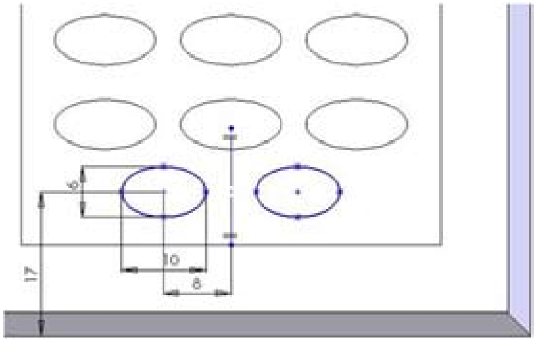
SolidWorks copies the entire pattern. If you change the original cut feature, all the copies are updated.

60. Rename the pattern feature **btn_row234**.

You can adjust the spacing, total instances, and direction in the patterns if needed. Right-click the pattern feature in the FeatureManager design tree, and select **Edit Feature**. Make the necessary adjustments, check the preview, and click **OK** when you are satisfied with the result.

Two More Buttons

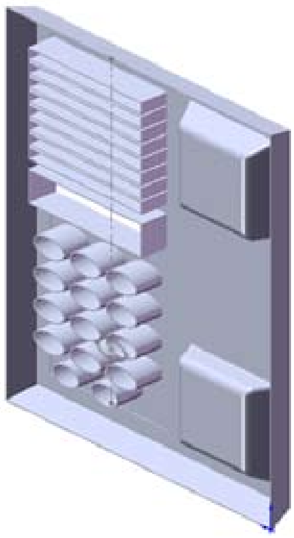
61. Open a sketch on the raised panel face. Sketch and dimension the ellipses as shown. If you want, add a vertical center-line (watch the V near the cursor) across the midpoint (eventually add a midpoint relation later). Use mirroring after you sketched the first ellipse.



62. Extrude the sketch as a through all cut.

63. Rename the feature **2_low_btns**.

Shelling the Part



64. Click **View, Orientation**, and double-click **Back**.

65. Click the back face of the part to select it.

66. Click **Insert, Features, Shell**.

67. In the **Shell1** PropertyManager, set the **Thickness** to **0.5 mm**.

68. Click **OK**.

Reordering Features

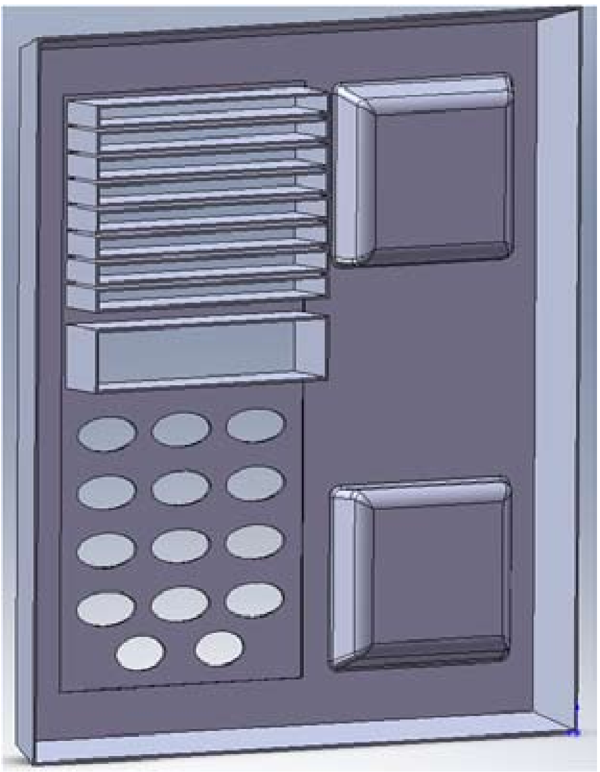
SolidWorks builds the features of the part in the sequence shown in the FeatureManager design tree (the order in which you created them). Therefore, changing the order of the features can change the characteristics of the features.

Maybe it was a mistake to cut the button holes before shelling the part. If you want the button holes to extend only through the shell thickness (not through the full depth of the part), you can fix it by taking the steps below.

69. In the FeatureManager design tree, select the **Shell** feature and drag it into position before the **button** feature. The part rebuilds itself.

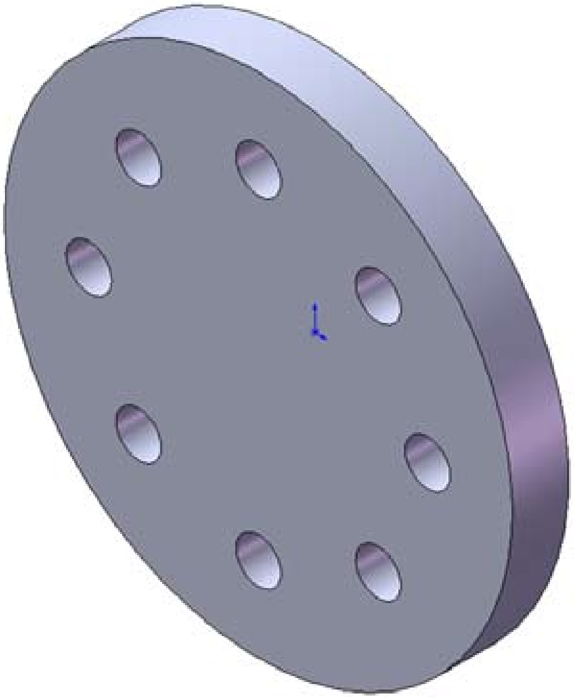
70. Rotate the model with the cursor or with the Arrow keys. The button holes now are openings through only the front face of the part. Notice the difference between cuts made before and after the shell operation.

71. Save and close the part after noticing the difference mentioned above.



Creating a Circular Pattern

This section covers the feature circular pattern. You will learn to create the model below.



Creating the Base and the Feature to Pattern

1. Open a new part, and open a sketch.

2. Click **Tools, Options**. On the **Grid/Units** tab, make sure that the **Length Unit** is set to **millimeters**, and click **Snap to Points** off.

3. Sketch a circle of **90 mm** in diameter with its center at the origin of the sketch.

4. Click or **Insert, Base, Extrude**, and extrude the profile **10 mm**.

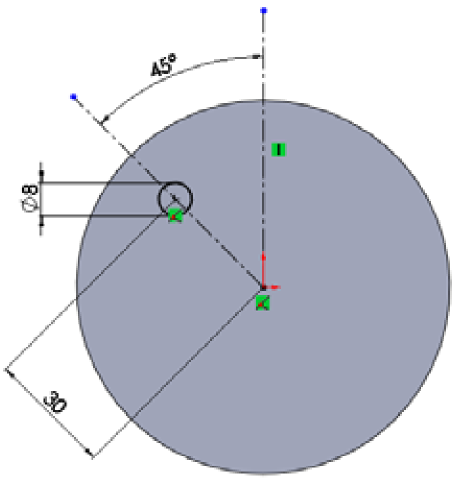
5. Set the **View Orientation** to **Front**.

6. Click on the front face of the part and open a new sketch.

7. Click **Centerline** or **Tools, Sketch Entity, Centerline**, and sketch a vertical and a diagonal centerline from the origin outwards.

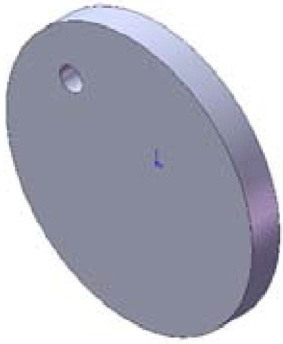
8. Dimension the centerlines to a **45°** angle. To place an angular dimension, select the two lines and place the dimension in between.

9. Sketch a small circle on the diagonal centerline and dimension with an **8 mm** diameter and a a **30 mm** distance to the origin.



10. Click or **Insert, Cut, Extrude**, and extrude a **Through All** cut.

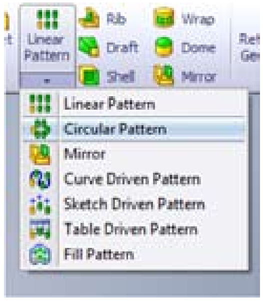
    This is the original instance of the feature to repeat in a circular pattern around the disk.



11. Change view to **Isometric** to get a better view of the result.

Creating a Circular Pattern

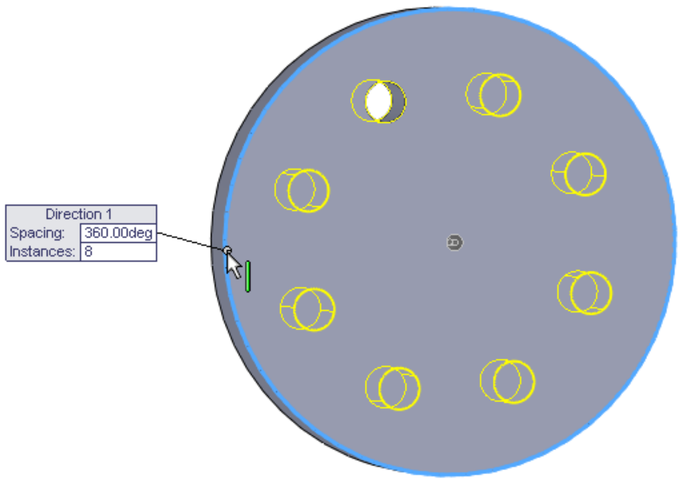
12. Click **Circular Pattern** , or **Insert, Pattern/Mirror, Circular Pattern**. The Circular Pattern PropertyManager appears.



13. The first parameter is the Pattern Axis, around which the pattern rotates. By selecting the **outside edge** of the current face, its centerpoint will be used to rotate the pattern around. You can also select the **cylindrical face**.

    Tip: another possibility is using a (center)line perpendicular to the pattern rotation as the Pattern Axis. this could be a line created in another sketch. This is useful when creating a circular pattern without having a corresponding cylindrical edge or face.

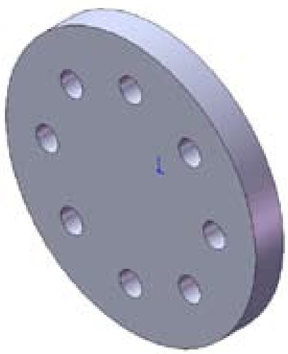
14. Click in the **Features to pattern** list to activate it (highlighting it in blue). Select the **Cut-Extrude** in the FeatureManager design tree (which you can access by using the + in the top-left corner).



15. When setting the number of instances by the pattern, there are two possibilities:
    - Defining the **Angle** between **each** instance:
        - Set the **Angle** to **45°**.
        - Set the **Number of Instances** to **8**.
    - Dividing the **Total Angle**, spacing the instances equally:
        - Set the **Total Angle** to **360°**.
        - Check the **Equal Spacing** box.
        - Set the **Number of Instances** to **8**.

16. Try both methods, watching the yellow preview carefully. Click **OK** afterwards.

    SolidWorks creates the pattern of cuts around the part center.



Using an Equation in the Pattern

You can use an equation to drive the hole pattern. In this example, the equation calculates the spacing angle by dividing 360° by the number of holes desired. This creates a full circle of equally spaced holes.

Important: *This feature cannot be used in combination with the* ***equal spacing*** *option. So, before continuing, make sure you used the* ***first*** *method of* ***step 15****.*

17. In the FeatureManager design tree, double-click the **CirPattern**. Two values appear on the part: total instances (8) and spacing angle (45°).

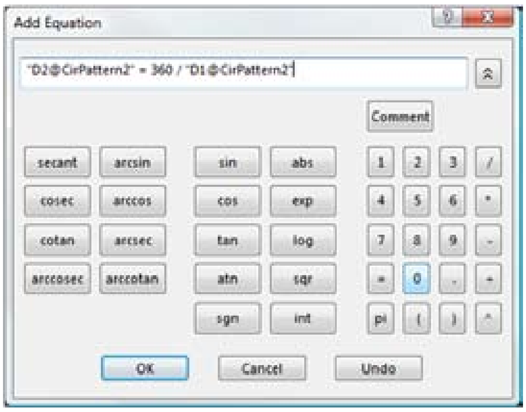
18. Click **Tools, Equations** and click **Add** in the **Equations** dialog box.

19. Click the **spacing angle dimension** on the part (You may have to move the dialog boxes to uncover it.) Its name (**D2@ CirPattern**, the second dimension in the circular pattern) is entered in the text field of the **Add Equation** dialog box.

20. Using the calculator buttons in the **New Equation** box, enter = **360 /** (or type **=360/** on the keyboard).

21. Click the total instances value. **D1@CirPattern2** is added to the equation.

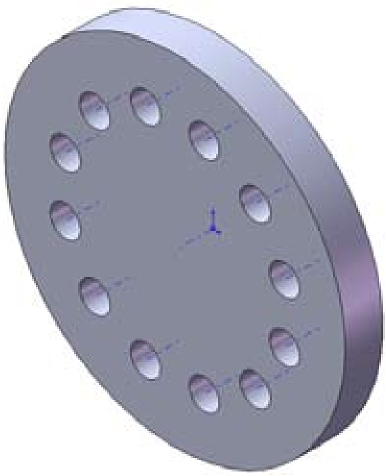
22. Click **OK** to complete the equation, and click **OK** again to close the **Equations** dialog box.



An **Equations**  folder is added to the FeatureManager design tree. To add, delete, or edit an equation, right-click the folder, and select the desired operation.

23. Now test the equation by changing the number of holes in the pattern:
   - Double-click the total instances value (8).
   - Set the value in the spin box to the number of holes that you want, in our case, 12.

24. Click **Rebuild**  in the **Modify** dialog box (or press **Enter**, then click **Rebuild** on the Standard toolbar).



Skipping Instances in a Pattern

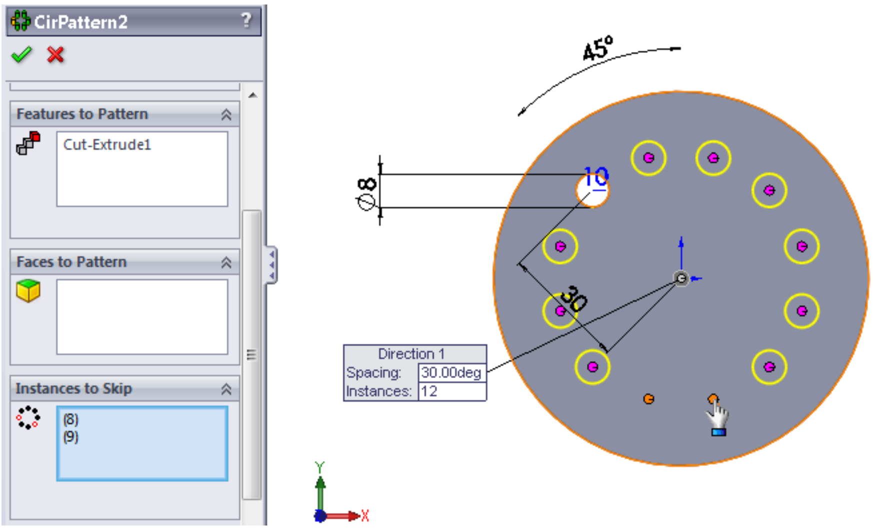
Sometimes, you may want to skip one or more of the instances of your pattern. You can achieve this by using the **Instances to Skip** option.

25. Right-click the **CirPattern** and choose **Edit Feature**.

26. In the PropertyManager, scroll down and expand the **Instances to Skip** box. Notice that the centerpoints of each instance appears.

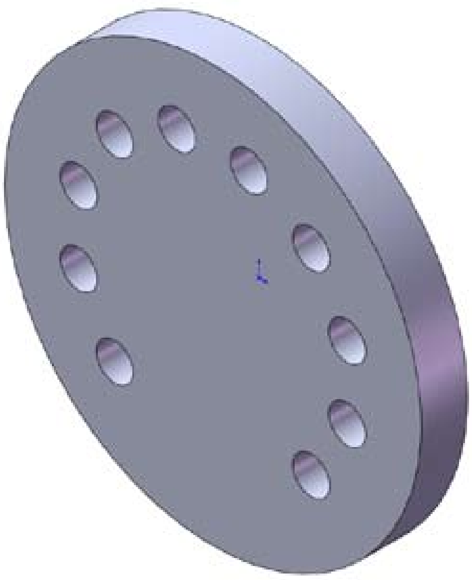
27. For a better view, change it to **Front** view.

28. We want to skip the bottom two instances. Select their centerpoints. Notice that the cursor changes into a hand.



29. Click **OK**.

30. **Save** the part in */work directory/* as **pattern_part.SLDPRT**.

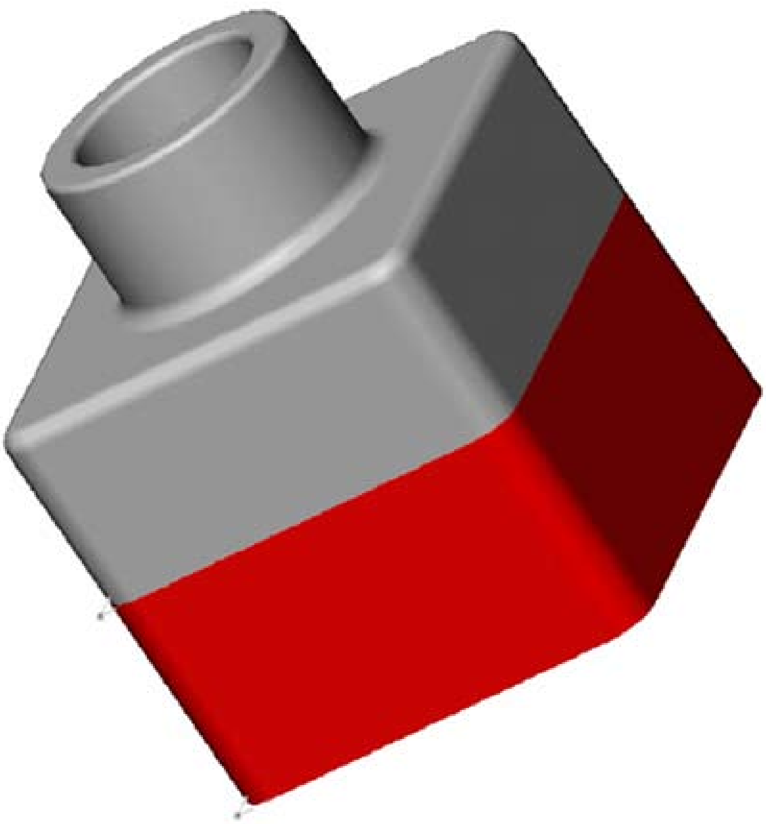


Chapter 3: Assembly Modelling

In this chapter it will be explained what an assembly is. By means of examples the making of assemblies will be explained and it will be clarified which features can be used.

Assembly Basics

An assembly is a set of parts. The amount of parts can differ from two to hundred or more. An assembly can also be composed of one or more other assemblies, which are called sub-assemblies. In this chapter you will learn to build a simple assembly. A new base part will be created and will be related to the part made in the ´Part Basics´ paragraph of chapter 2 to create an assembly.



Creating the Base Feature

You can use the same methods you learned in Chapter 2, Part, to create the base fore a new part.

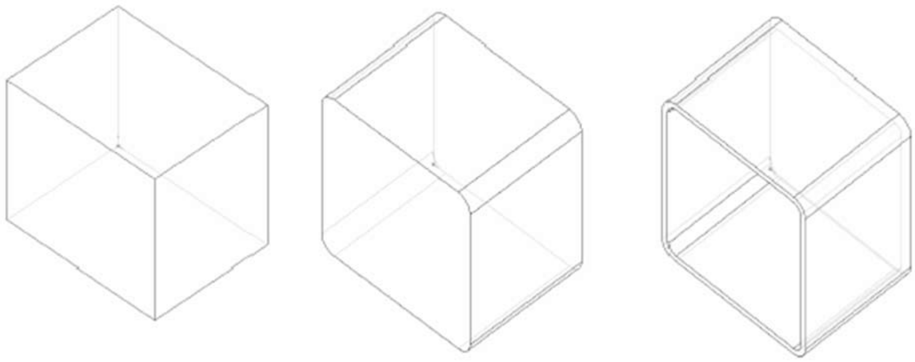
1. Click **New** on the Standard toolbar, or click **File, New** on the Menu bar and create a new part document.

2. Open a **Sketch** , and sketch a rectangle beginning at the origin.

3. **Dimension** the rectangle to **120 mm** x **120 mm**.

4. **Extrude Boss/Base** the rectangle as a Solid Feature, with a **Blind End Condition**, to a **Depth** of **90 mm**.

5. **Fillet/Round** the four edges shown with a radius of **10 mm**.



6. **Shell** the front face of the part to a **Thickness** of **4 mm**.

7. **Save** the part in your */work directory/* as **TUTOR2** (SolidWorks adds the **.SLDPRT** extension).

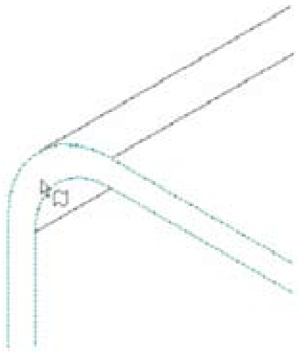
Creating a lip on the part

In this section, you use the Convert Entities and Offset Entities tools to create sketch geometry. Then a cut creates a lip to mate with the part from Chapter 3, Part.

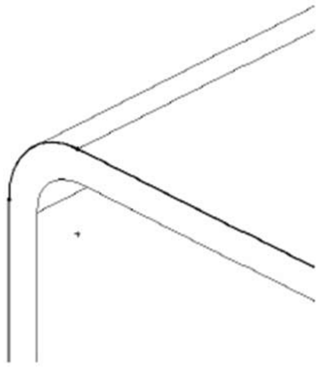
Tip: *Using the Selection Filter makes it easier to select the faces in this section. Right-click in the toolbar area, and check **Selection** Filter in the drop-down list. Then click **Filter Faces** . When you are finished with this section, click **Filter Faces** again, or click **Clear All Filters** .*

8. Zoom in on a corner of the part, select the thin wall on the front face of the part, and click **Sketch** to open a sketch.

9. The front face is still highlighted. If this is not the case, click the front face again. The edges of the part face are highlighted.



10. Click **Convert Entities** on the Sketch Tools toolbar or **Tools, Sketch Tools, Convert Entities**. The outer edges of the selected face are projected (copied) onto the sketch plane as lines and arcs.

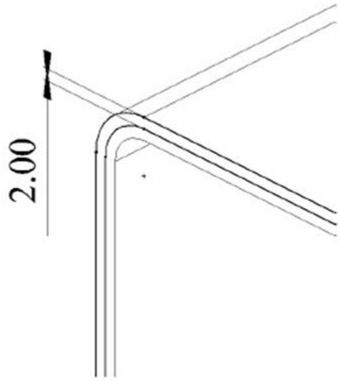


11. Click the front face again.

12. Click **Offset Entities** on the Sketch Tools toolbar or **Tools, Sketch Tools, Offset Entities**. The **Offset Entities** dialog box appears.

13. Set the **Offset Distance** to **2 mm**. The preview shows the offset extending outward.

    Select the **Reverse** check box to change the offset direction.



14. Click **OK**.

    A set of lines is added in the sketch, offset from the outside edge of the selected face by 2 mm. This relation is maintained if the original edges change.

15. Click or **Insert, Cut, Extrude**.

16. In the **Extrude Feature** dialog, set the **Depth** to **30 mm**, and click **OK**.

    The material between the two lines is cut, creating the lip.

Changing the Material of a Part

You can add a **Material** to a Part. This will define the technical properties of the part used for later analysis, such as its density for calculating its weight.

17. Right-click on the Part in the FeatureManager and select **Material, Edit Material**. The **Material** dialog appears.

18. Scroll down and choose **ABS**, under **Plastics** as the Material. The properties of this Material are now shown to the right.

19. Under the **Appearance** tab, deselect **Applt appearance of: ABS**. We will add this ourselves in the next step.

Changing the Color of a Part

You can change the color and appearance of a part or its features.

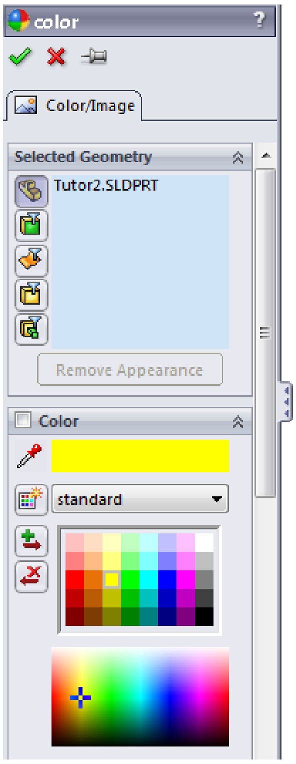
20. Select **Shaded** as Display Style.

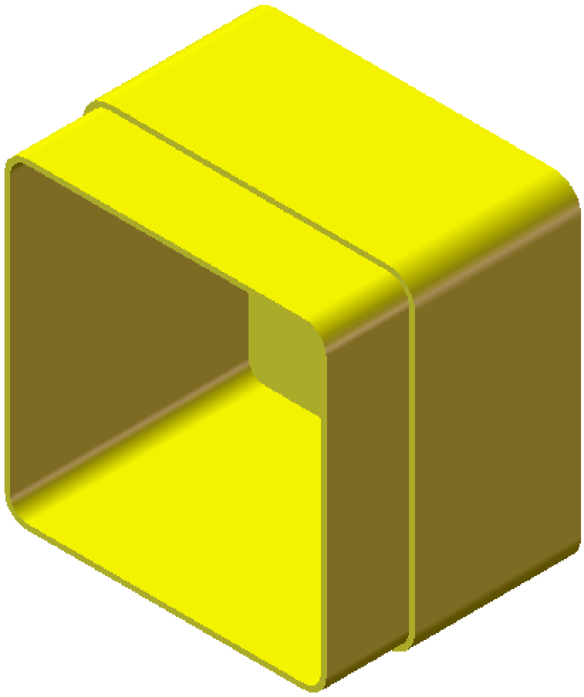
21. Make sure the part is deselected and click **Edit Appearance** on the View Toolbar. The **Color** dialog box appears.

    *Note: Possibly the **Appearance/Scenes** tab of the Task Pane opens up to the right. Simply click anywhere in the graphics area to close it.*

22. Click a different color than gray on the palette, then click **OK**.

23. Save the part.





Creating the Assembly

Now you can create an assembly using the two parts, **TUTOR1** and **TUTOR2**.

24. If **TUTOR1.SLDPRT** (from Chapter 2) is not still open, click **Open** on the Standard toolbar and open it from your */work directory/*.

25. Click **File, New** on the Standard toolbar, select **Assembly** and click **OK**. The **Begin Assembly** PropertyManager appears. We will add the parts ourselves in the next steps, so click **Cancel** .

26. Click **Window, Tile Horizontally** to display all three windows. Close any extra windows.

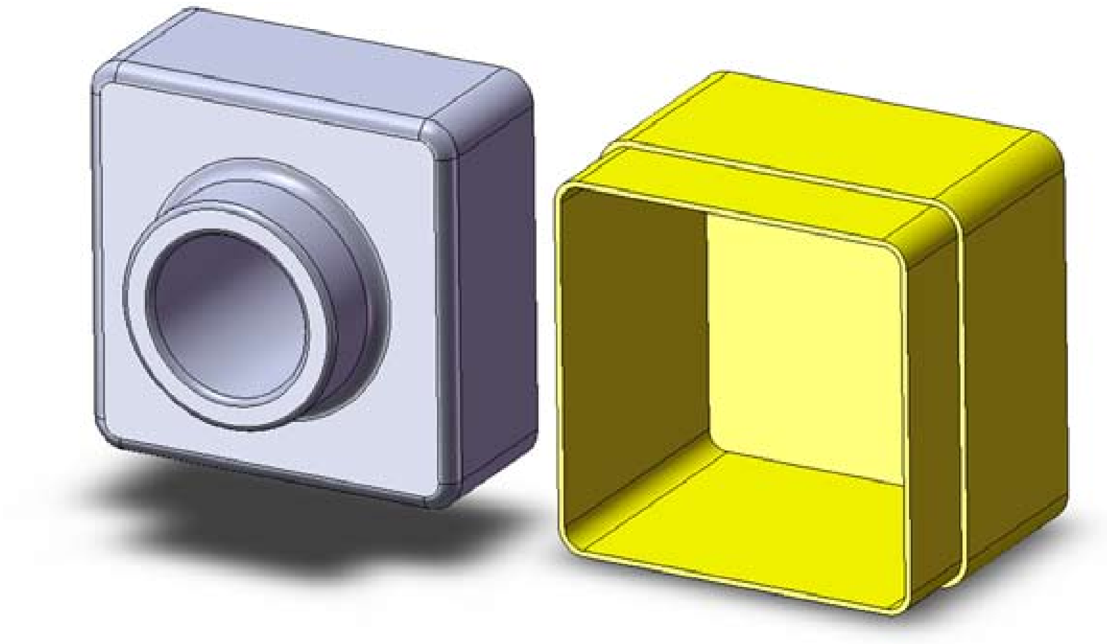
27. Drag the **TUTOR1** icon from the top of the FeatureManager design tree for **TUTOR1.SLDPRT**, and drop it in the FeatureManager design tree of the assembly window (**Assem1**).

Notice that as you move the pointer into the FeatureManager design tree, the pointer changes to .

Adding a part to an assembly this way results in the part's origin coinciding with the assembly origin. The planes of the part are also automatically aligned with the planes of the assembly and the part will be fixed in the 3D space of the assembly. This means it is fully defined.

28. Drag the **TUTOR2** icon from **TUTOR2.SLDPRT**, and drop it in the **graphics area** of the assembly window, beside the **TUTOR1** part.

    Notice that as you move the pointer into the graphics area, the pointer changes to .



29. Save the assembly in your */work directory/* as **TUTOR** (SolidWorks adds the **.SLDASM** extension). If you see a message about saving referenced documents, click Yes.

30. Drag a corner of the assembly window to enlarge it, or click the enlarge window icon in the upper right corner to make the window full size. You no longer need to have the **TUTOR1.SLDPRT** and **TUTOR2.SLDPRT** windows in view.

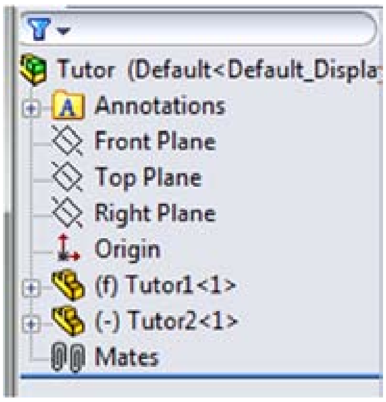
31. Click **Zoom to Fit** or press "**f**" on your keyboard.

    Note: *Using* ***Insert, Component, Existing Part/Assembly*** *from the menu bar or is an other way to add existing parts to an assembly.*

Manipulating the Components

When you add a part to an assembly, it is referred to as a component of the assembly. You can move or rotate the components individually or together using the tools on the Assembly toolbar.

The first component you add to an assembly is fixed in place by default. A fixed component has the prefix **(f)** in the FeatureManager design tree. You cannot move it or rotate it unless you float (unfix) it first. This also means that the reference planes of the component match the planes of the assembly, and the component is fully defined. Right-click the component with the closed bottom (**Tutor2**) in either the FeatureManager tree or in the graphics window, and then select **Fix** from the rightmouse menu. The prefix changes to (f), indicating that the component's position is fixed. Moving this part will fail. Switch back to floating, by right-click, **Float**.



Consider assembling a washing machine. The first component logically would be the frame on to which everything else is mounted. By aligning this component with the assembly's reference planes, we would establish what could be called "product space". Automotive manufacturers refer to this as "vehicle space". This space creates a logical framework for positioning all the other components in their proper positions.

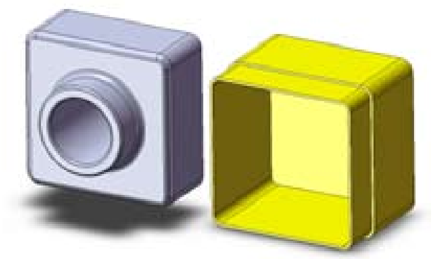
In the following section, you can practice moving and rotating components in the assembly.

32. Select the **TUTOR2** component. You can either select its name in the FeatureManager tree, or select one of the component faces. Because SolidWorks has probably placed the two parts in the same place, the first option is more convenient here.

33. Click one of these tools:
    - **Move Component**,
    - **Rotate Component**.

34. Move or rotate the components as desired. Be sure to conclude your move action with the parts positioned more or less as shown below.

35. To exit from a move or rotate mode, you can:
    - Click **Escape**.
    - Click **Tools, Select**.



If you only want to change the view orientation or location of the components, use the **View Orientation** box, and the **Pan** and **Rotate View** buttons on the View toolbar.

Mating the Components

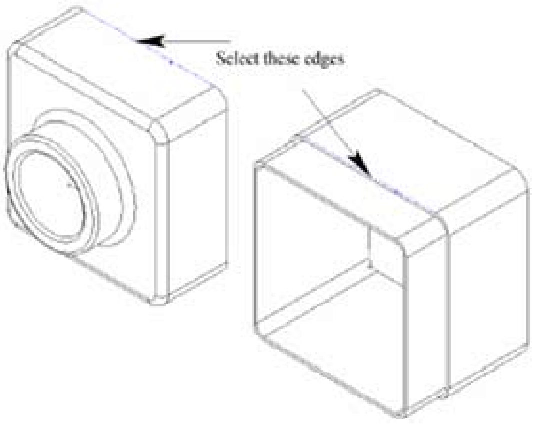
In this section, you define assembly mating relations between the components, making them align and fit together.

36. Select **Isometric** as **View Orientation** from the View toolbar.

37. Click **Mate** on the Assembly toolbar, or click **Insert, Mate.**
    The **Mate** PropertyManager appears.

38. Click the top edge of **TUTOR1**, and then click the outside edge of the lip on the top of **TUTOR2**.

    The edges appear in the **Entities to Mate** list.



39. If not automatically selected, select **Coincident** under **Standard Mate**, and **Aligned** under **Mate Alignment**.

40. Select **Show preview** to preview the mate.

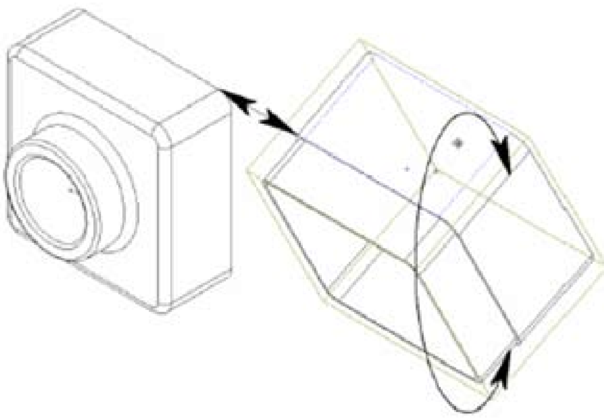
    The selected edges of the two components are made coincident. Also the **Alignment** Condition is set to **Aligned** .

41. Click **OK**.

42. Click **OK** again to close the **Mate** PropertyManager.

    The positions of the components in the assembly are not yet fully defined, as shown by the (-) prefix in the FeatureManager design tree. They still have some degrees of freedom to move in directions that are not yet constrained by mating relations.

43. Click the **TUTOR2** component, then click **Move Component** . Notice the cursor shape . Drag the component from side to side.



44. Select **TUTOR2**, hold the **Ctrl** key, select the mated edge, and click **Rotate Component**. Under **Rotate** select **About Entity.** The edge appears as the entity around which will be rotated. Drag to rotate the component around the mated edge.

Adding More Mates

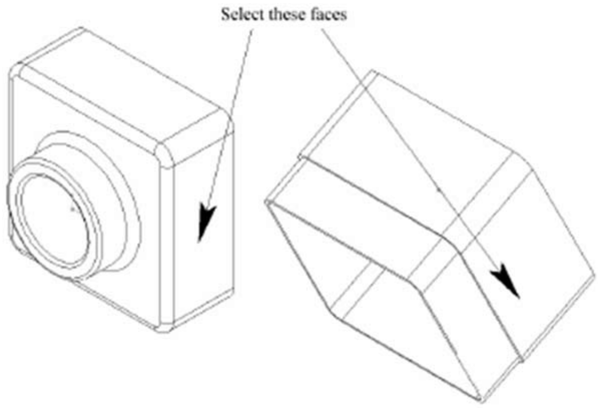
45. Select the right hand face of one component, then Ctrl-select the corresponding face on the other component.

46. Click **Mate** , or click **Insert, Mate**.

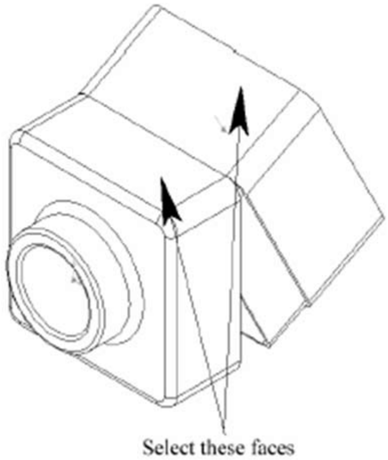
47. In the **Mate** Property Manager, select **Coincident** .

48. Click **Preview** to preview the mate. The components should mate as shown below.

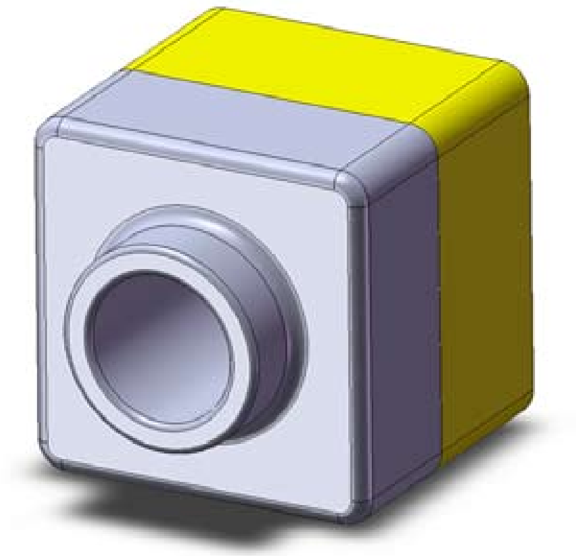
49. Click **OK**.



50. Add a third **Mate** by selecting the top faces of both components, and choosing another **Coincident** mate. The part **TUTOR2** will lose its under defined minus sign.

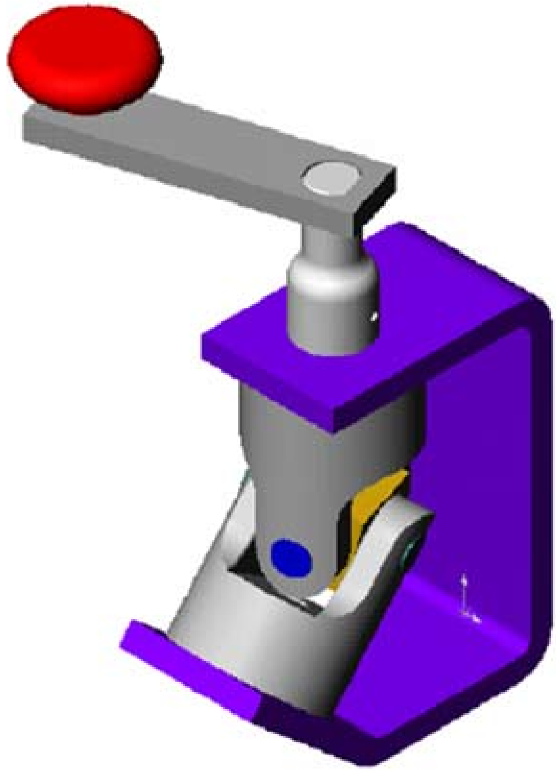


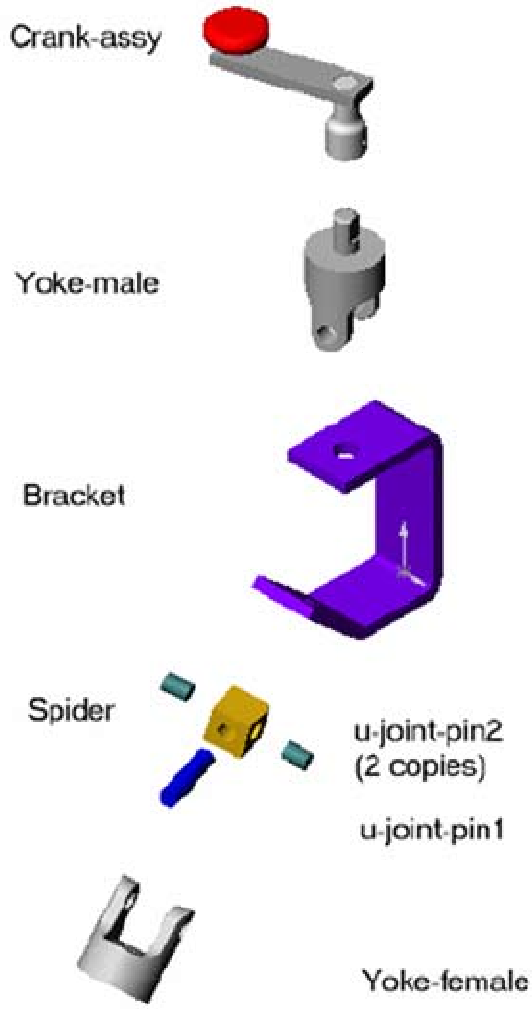
51. Save the assembly (if the program suggests a rebuild, click **Yes**). Close the part.



Advanced Assemblies

After successfully completing this part you will be able to create the model shown below.





Creating a New Assembly

1. Open a new assembly using the **New Document** icon. Click the **Assembly** and **OK** to create a new assembly.

2. Using **Tools, Options**, check if the units on the document properties tab are set to **MMGS**. The assembly units can be different from the parts units. You can assemble inch and millimeter parts into the same assembly. However, when you edit the dimensions of any of the parts in the context of the assembly, they will be displayed in the units of the assembly, not those of the part itself.

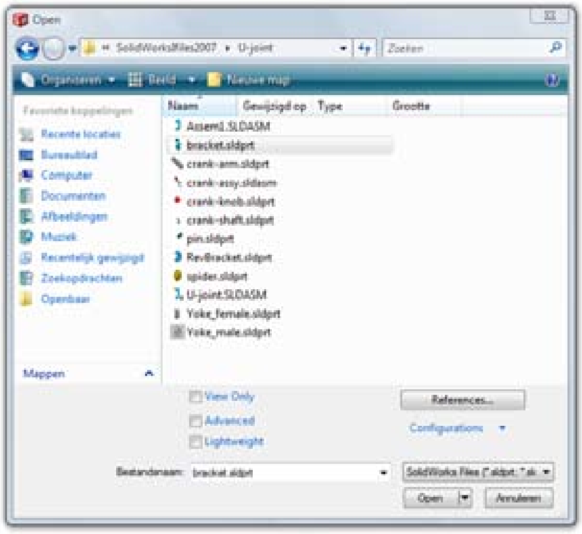
3. If needed, create a new **template** for assemblies. Use the same technique as used for creating new document templates as explained in **Chapter 1**, **steps 5-16** of the first paragraph ("Controleren van de instellingen").

Adding the First Component

The first component added to the assembly should be a part that will not move as is explained in the previous part of this chapter. By fixing the first component, others can be mated to it without any danger of it moving. There are several ways to add components to the assembly:

- use the **Insert** menu,
- drag them from an open document,
- drag them from the **Explorer**.
- Use **Insert Component** from the CommandManager.

4. Use **Insert Component** . The **Insert Component** Property-Manager appears. **Browse** for **'Bracket.sldprt'** from the **'Chapter 3/U-joint'** map.



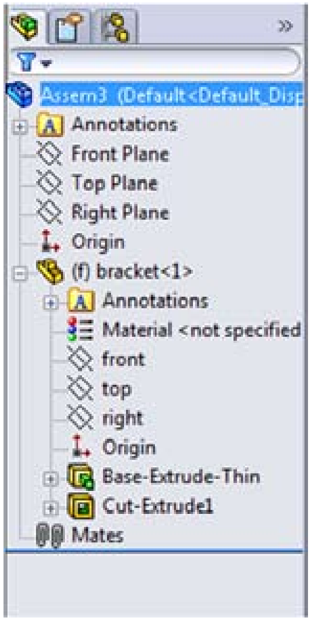
5. Click **OK**, and the component is dropped at the origin in the assembly. The part will appear in the assembly FeatureManager design tree as **Fixed (f)**.

FeatureManager Design Tree

Within the FeatureManager Design Tree of an assembly, the folders and symbols are slightly different than in a part. There are also some terms that are unique to the assembly. Now that some parts and mates are listed there, they will be described. An example of the FeatureManager Design Tree is shown on the next page.

Just like parts, the assembly has an **origin** and **three reference planes**. If you can remember the "washing machine example" from the last paragraph, the first component can be related to this to create a logical framework for positioning all the other components in their proper positions.

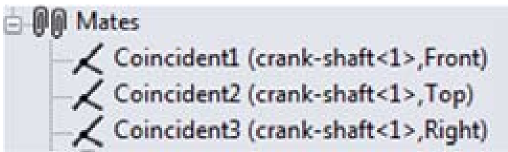
Next, you see the **components** of the assembly. These are parts made earlier, such as the bracket, that you want to combine. instead of parts, you can also insert other assemblies, making them sub-assemblies of the current one.



The following details are important about the components:

- **Component Part Folder**
  By clicking the plus sign + you can expand the **Component Part Folder**. This contains the **Design Tree** of that specific part, including all features, planes and axes.
- **Instance Number**
  The instance number indicates how many copies of a certain component part are found in the assembly. The **<1>** of the name **bracket<1>** indicates that this is the first instance of the bracket.
- **State of the component**
  Components are positioned using **mates**. The **State of the component** shows if the position of the part is completely defined.
  Defining component positions is similar to defining a sketch, only instead of **Smart Dimensions**, you use **mates**. So a component position can also be **Fully**, **Under** or **Over Defined**. If the position isn´t **Fully Defined**, A **(+)** or **(-)** sign will be in front of the name, showing it is **Over** or **Under Defined** respectively. Parts that are under defined have some degrees of freedom available.
  There are also two other possible **States**. The **Fixed State (f)** indicates a component is fixed in its current position, but not mated. The question mark **(?)** is for components that are **Not Solved**. This means there is an error and SolidWorks cannot place the part using the information given.

Lastly, the **Mate** Folder shows all the mates of the assembly, used to position the components. Mates can be used to fully define a component that does not move, or under define one that is intended to move. Expand the folder by clicking the plus sign + . Moving the cursor over the mates highlights the Entities used by the mate. More will be explained later on.



For more information, refer to the **SolidWorks Online Help**.

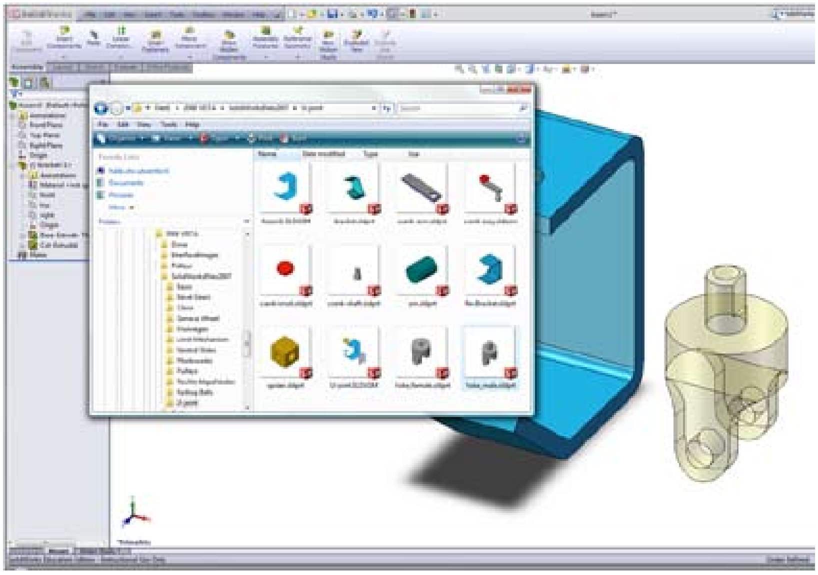
Mating Components to Each Other

Once the first component has been inserted and fully defined, other parts can be added and mated to it. In this example, the **Yoke_male** part will be inserted and mated. This part should be under defined so that it is free to rotate.

The next component connects to the bracket, but not to the assembly planes directly. All other components will mate with other components, not the assembly itself.

6. Open the **Explorer** window and size the window so the graphics area of SolidWorks is still partly visible. Since SolidWorks is a native Windows application, it supports standard Windows techniques like **'drag and drop'**. The part files can be drag-copied from the Explorer window into the assembly to add them.

7. Drag and drop the **'Yoke_male.sldprt'** into the graphics area from the same directory as the **bracket**.



The new component is listed as: **(-) Yoke_male <1>**. This means that the component is the **first instance** of **Yoke_male** and it is **Under Defined**. It still has all degrees of freedom.

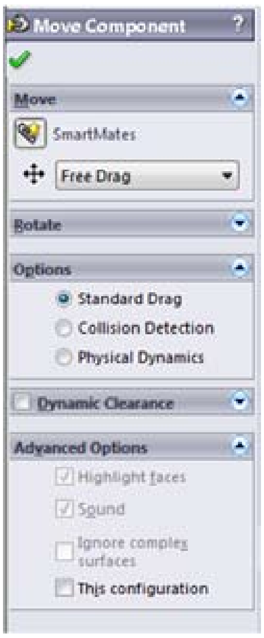
Move and Rotate Component

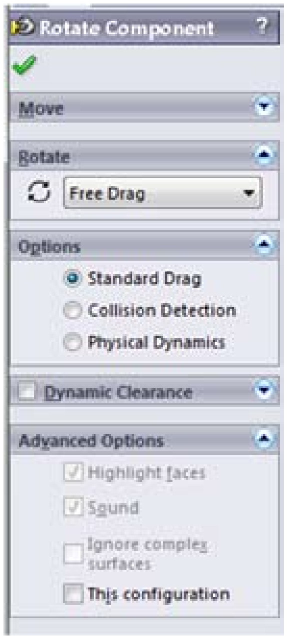
In the first paragraph of this chapter the move and rotate commands were introduced. More possibilities of the commands will be discussed below.

As was already discussed, one or more selected components can be moved or rotated to reposition them for mating using the **Move** and **Rotate Component** commands. Also, moving under defined components simulates movement of a mechanism through dynamic assembly motion.

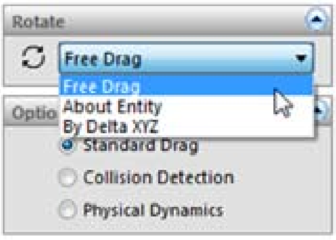
8. Select the **Yoke_male**. Click **Move component** .

Move Component and **Rotate Component** behave as a single, unified command. By expanding either the **Rotate** or **Move** option in the PropertyManager, you can switch between the two commands without ever closing the PropertyManager.



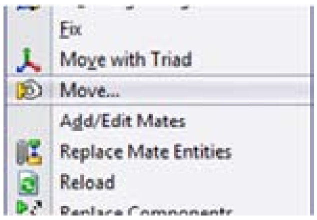


If you want to move the component in a more specific way, The **Move** tool has several options for defining the type of movement. The option **Along Entity** has a selection box, **Along Assembly XYZ**, **By Delta XYZ**, and **To XYZ Position** require coordinate values.

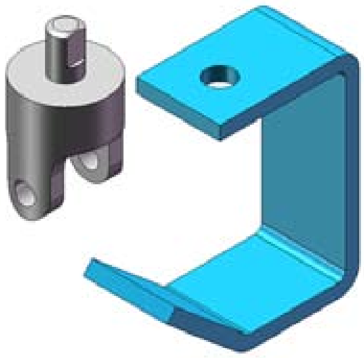


The **Rotate** tool also has the options **About Entity** and also **By Delta XYZ** to define how the component will rotate.

Another option is to use the right-click menu. You can easily switch between moving and rotating a component by right-clicking in the graphics area, and selecting the desired function from the shortcut menu.



9. Reposition the **Yoke_male** on the screen so it is easy to work with.



Mate to another Component

Obviously dragging a component is not sufficiently precise for building an assembly. You can use faces and edges to mate components to each other as you have seen in a previous part of this chapter. To practice mating, some parts will be placed inside the part **Bracket**. The parts which will be placed inside Bracket are intended to move, so make sure that the proper degree of freedom is left available.

Insert Mate creates relationships between component parts or between a part and an assembly. Two of the most commonly used mates are **Coincident** and **Concentric**. Mates are always made between a pair of objects.

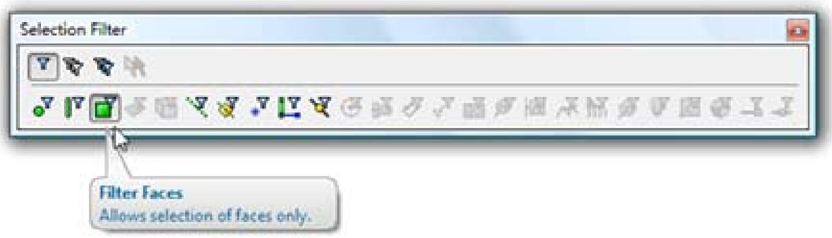
Mates can be created using many different objects. You can use:
- Faces
- Plane
- Edges
- Vertices
- Sketch lines
- Axes
- Origins

To insert a Mate, Click **Mate** on the CommandManager of choose: **Insert, Mate.**

The **Yoke_male** component needs to be mated so that its shaft aligns with the hole and the flat face contacts the bracket inner face. Concentric and Coincident mates will be used.

Selection filter

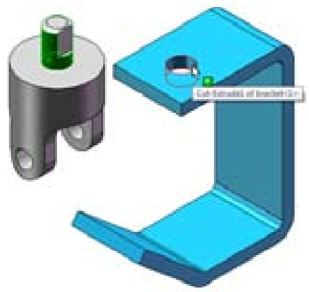
With the **selection filter**, you can set the type of items you want to select. The selection filter option is very useful in mating. Since many mates require face selections, it is convenient to set the **Select** option to **Faces**. The filter will remain in effect until SolidWorks or the part is exited, or until the filter is switched-off (by clicking or **Toggle Selection Filters**) or changed.



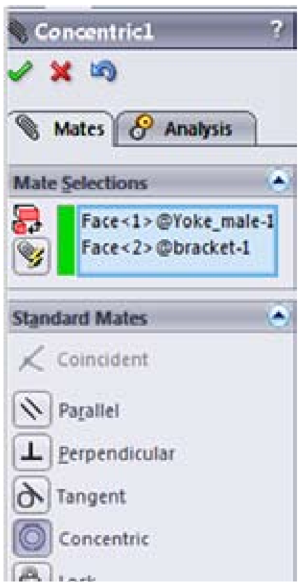
10. To display the Selection Filter toolbar, right-mouse click in the toolbar area, and check **Selection Filter**.

11. Turn on the **Filter Faces** by clicking .

12. Click on the **Insert Mate** icon to access the PropertyManager. If the dialog is open, the face selections can be done without using the **Ctrl**-key.



13. Select the faces of the Yoke_male and the bracket as indicated below by the dotted lines. If you select the faces before you open the dialog, you must pick the second face using the **Ctrl**-select method. That's why we recommend opening the dialog first.

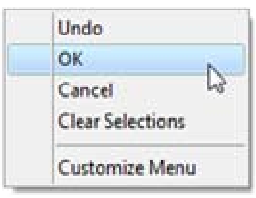


The faces are listed in the **Entities to Mate** list. Exactly two items should appear in the list. Only mate types that are valid for the selected geometry (**Parallel, Perpendicular, Tangent, Concentric, Distance** and **Angle**) are listed.

14. If not already selected, click **Concentric** .

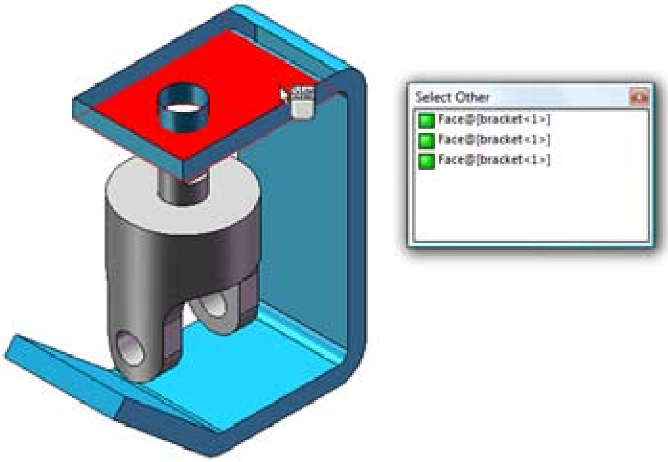
15. Click **OK** to add the mate.

Note: *The right mouse menu provides options to apply the command (**OK**) or **Cancel** it.*

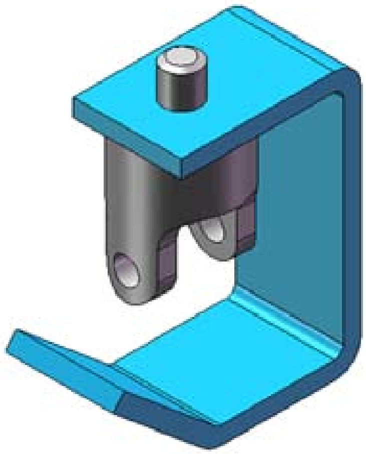


To select faces that are hidden or obscured, you can either re-orient the view, or you can use the **Select Other** option. When you position the cursor in the area of a face and press the right mouse button, **Select Other** is available as an option on the shortcut menu. When you choose that option, the system will show a dialog box with available faces near the first selected face. When you move towards the top of the bracket you'll see the hidden face.

16. Select the hidden face of the bracket.



17. Add the mating relationship **Coincident** between the hidden face and the top face of the **Yoke_male**.



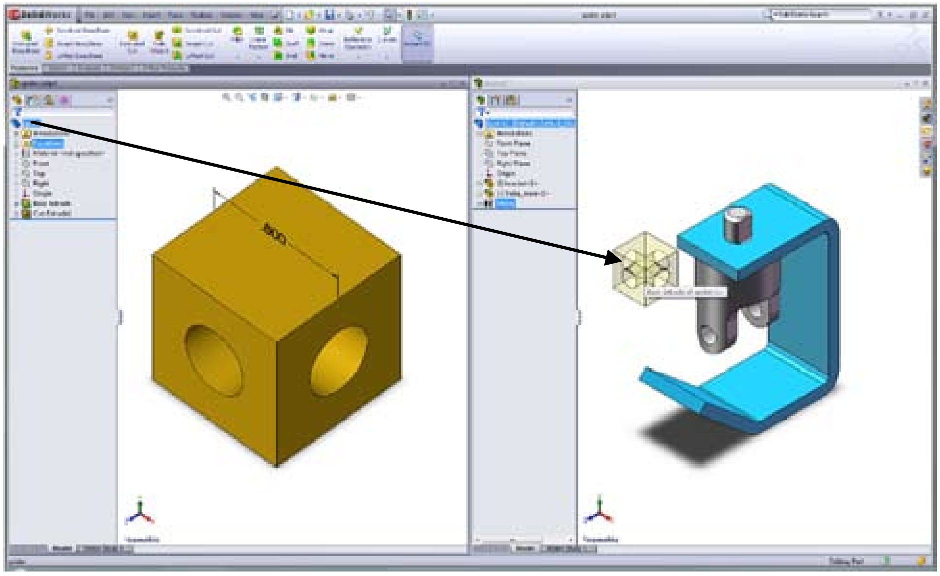
The Yoke_male component is listed as under constrained. It is still able to move by rotating around the axis of its cylindrical surface.

18. Test the behavior of the Yoke_male by using **Move Component** to rotate it.

A third part, a cube with two through holes named **'Spider'**, will be mated using **Coincident** and **Concentric** mates. We will insert Spider by dragging it in from an open document window.

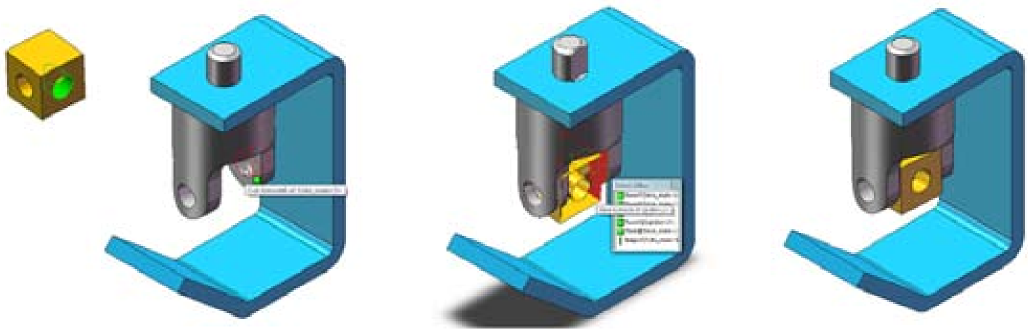
19. Open the part **'spider'**, again from the **'U-joint'** directory and tile the windows of the assembly and part vertically or horizontally.

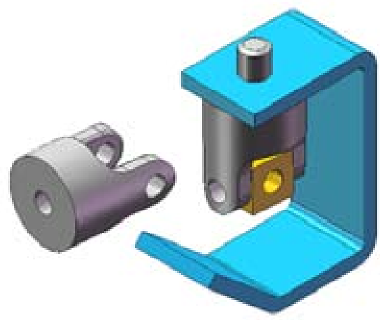
20. Drag the top level component of the spider into the assembly and drop it by releasing the mouse button.



21. Add two mates between the **spider** and the **Yoke_male**.
    - A **Concentric** mate between the two cylindrical faces.
    - A **Coincident** mate between the two planar faces.

    The results are shown below.





22. Add the **'Yoke_female'** component using the **Insert** menu, or by dragging it from the **Explorer** or open document.

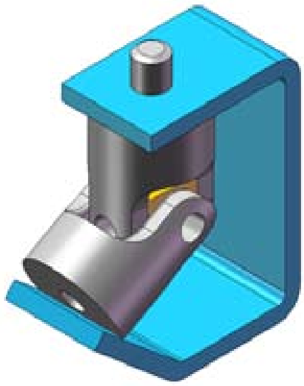
23. Drag and rotate the component into its approximate position.

    The flat face of the spider mates to the inner, flat face of the Yoke_female as **Coincident**. The cylindrical face in the spider mates to the cylindrical face in the Yoke_female as **Concentric**.

24. Select the **cylindrical** faces on the **spider** and the **Yoke_female** components.

25. Click **Concentric** and click **OK**.

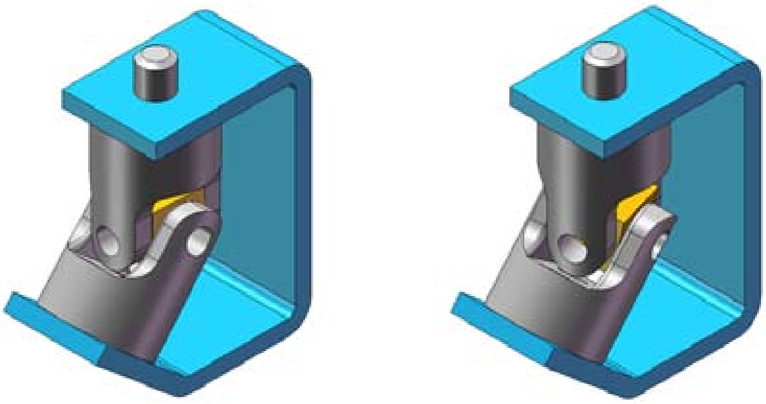
26. Do the same for the **Coincident** mate.



At this point you need to align the bottom face of the **Yoke_female** to the **angled face** of the **bracket**. You might be tempted to use a **Coincident** mate here but that would solve the mates incorrectly. Because of the clearance between the Yoke_female and the bracket, a **Coincident** mate is unsolvable. The gap prevents coincidence. Attempting a **Coincident** mate would over define the assembly. Instead, you should use a **Parallel** mate.

27. Using the **Mate** tool, add a **Parallel** mate between the flat face of the **Yoke_female** and the angled face of the **bracket**.

__Tip:__ Remember to use __Select Other__ to facilitate selecting hidden faces.



28. Select the Yoke_male and **Move Component**. Dragging the cursor forces the Yoke_male and mated components to turn.

__Note:__ The bracket component is fully defined and does not turn.

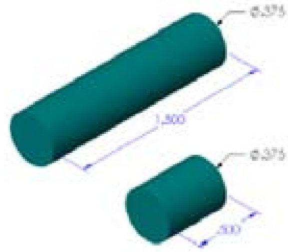
29. **Save** the assembly in */work directory/* as '**Universal Joint**'.

Using Part Configurations in Assemblies

Multiple instances of the same part can be used in an assembly, with each instance referencing a different configuration. We will use multiple instances of a part with different configurations in this assembly. There are two ways to create this type of configurations within a part:
- applying different dimension values to individual configurations,
- design tables (see the SolidWorks User's Guide).

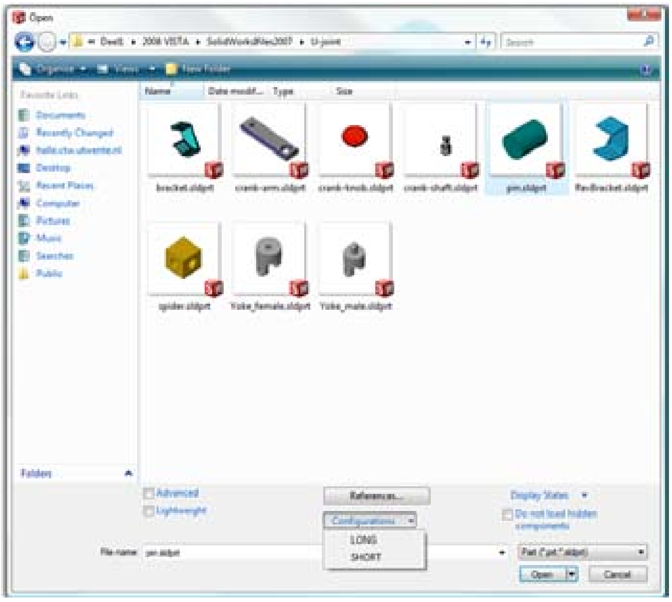
When you add a part to an assembly you can choose which of its configurations will be displayed. Or, once the part is inserted and mated, you can switch its configuration.

30. Open the part ‘**pin**’ from the same directory as bracket.sldprt.



The part pin has two configurations: **SHORT** and **LONG**. Which configuration was opened can be seen in the FeatureManager design tree behind the part name. The name of the configuration is shown there in capitals between brackets. Any configuration can be used in the assembly. In this case, two instances will use SHORT and one will use LONG.

So far we have covered three ways to insert a component into an assembly. You can use the command **Insert, Component, From File**, drag and drop from **Explorer** or drag and drop from the open document. We will examine these methods with respect to configurations.



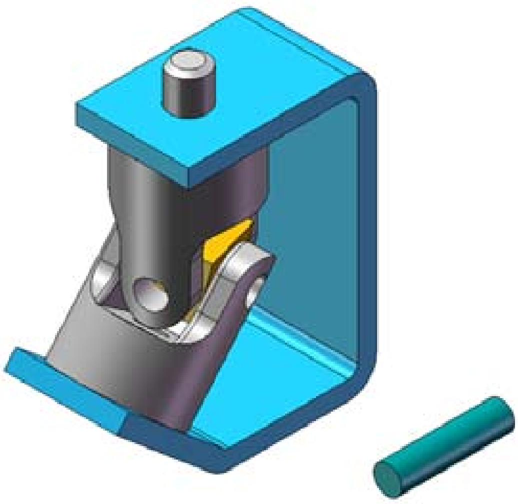
31. Close the part **pin**.

32. Click **Insert, Component, Existing Part/Assembly** and click **once** on the file **pin** from the browser.

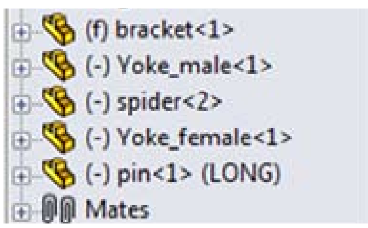
Below the button ‘references’ a button is displayed titled **Configurations** containing the two configurations of **pin**.

33. Select **LONG** in the **Configurations** box and click **Open**.

34. Click somewhere in the drawing window to place the component in the assembly.

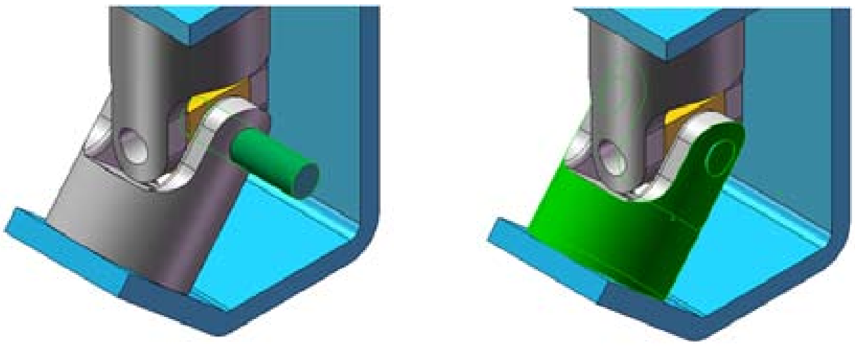


The added component appears in the FeatureManager design tree. The configuration used, in this case LONG, is appended to the component name.



35. Add a **Concentric** mate between the **pin** and the hole of **Yoke_female**.

36. Complete the mating using a **Tangent** mate between the faces indicated below.



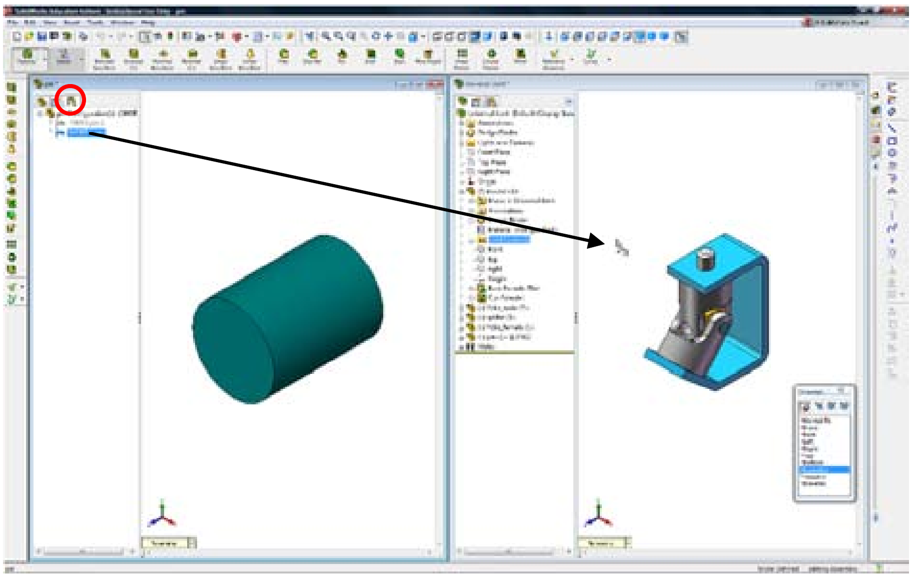
Another instance of the pin is needed. This one will be the shorter version, SHORT. We will open the pin, tile the windows of the part and assembly, and show the part's configuration menu.

When you need to access a component while working in an assembly, you can open it directly, without having to use the **File, Open** menu. The component can either be a part or a sub-assembly.

37. Right-click the **pin** component, either in the FeatureManager design tree or the graphics window, and select **Open Part**.

38. **Tile** the part and assembly **windows**.

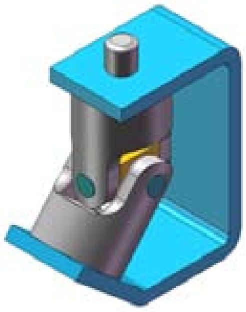
39. **Switch** to the **ConfigurationManager** of the **pin**.



40. **Drag and drop** the configuration **SHORT** into the graphics window of the assembly. You can drag and drop any configuration from the ConfigurationManager, not just the active one.

The component is added and it displays the proper configuration name in the FeatureManager design tree.

41. Mate the component using **Concentric** and **Tangent** mates.



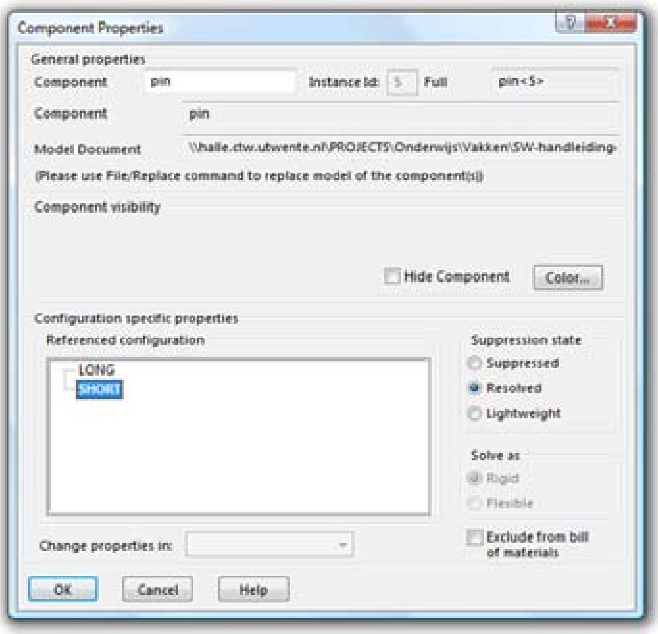
Many times parts and sub-assemblies are used more than once in an assembly. To create multiple instances, or copies of the components, copy and paste existing ones into the assembly.

42. Close the pin window and maximize the assembly window.

43. Create another copy of the pin component by holding the **Ctrl**-key while dragging the instance with the **SHORT** configuration from the FeatureManager design tree of the assembly. You can also ctrl-drag a copy by selecting the component in the graphics area.

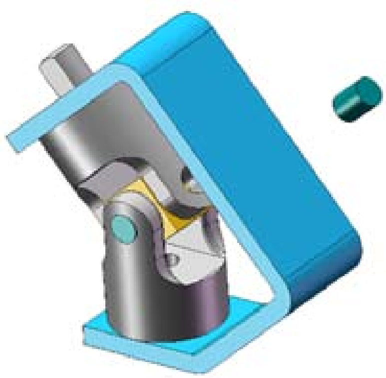
The result is another instance that uses the SHORT configuration, since it was copied from a component with that configuration.

44. Select the **pin<3>** component and choose **Properties**  from the right mouse menu.
The **Use named configuration** option is checked and set to SHORT.


Component Properties
General properties
Component pin Instance Id: 5 Full pin<5>
Component pin
Model Document \\halle.ctw.utwente.nl\PROJECTS\Onderwijs\Vakken\SW-handleiding
(Please use File/Replace command to replace model of the component(s))
Component visibility
Hide Component Color...
Configuration specific properties
Referenced configuration
LONG
SHORT
Suppression state
Suppressed
Resolved
Lightweight
Solve as
Rigid
Flexible
Change properties in:
Exclude from bill of materials
OK Cancel Help

This list can be used to change the configuration and to suppress or hide an instance. If **Referenced configuration** is set to **Use component's "in-use" or last saved configuration**, the saved configuration will be displayed.

45. Click **Cancel**.



46. Using the **Shift** and up-arrow keys, rotate the view twice at 90° intervals.

Hiding a Component

Hiding a component removes the component's graphics temporarily but leaves the component active within the assembly. A hidden component still resides in memory, still has its mates solved, and is still considered in operations like mass property calculations.

Hide Component turns off the display of a component, making it easier to see other parts of the assembly. When a component is hidden, its icon in the FeatureManager design tree appears in outline form like this: 

. **Show Component** turns the display back on.

Hide Component can be found:

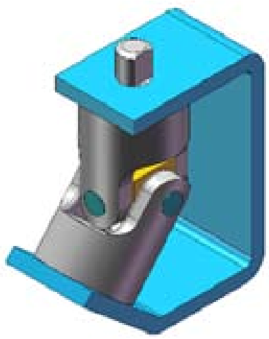
- Right-click the component, and click **Hide Component**  .
- From the pull-down menu, choose **Edit**, **Hide, Current Display State**.

47. Click on the bracket component and hide it using the toolbar option **Hide Component**  .

48. Mate the pin<3> component using the same types of mates used for the first one.

49. Unhide the bracket by selecting it and clicking the  icon again.

50. Return to the **Isometric** view.

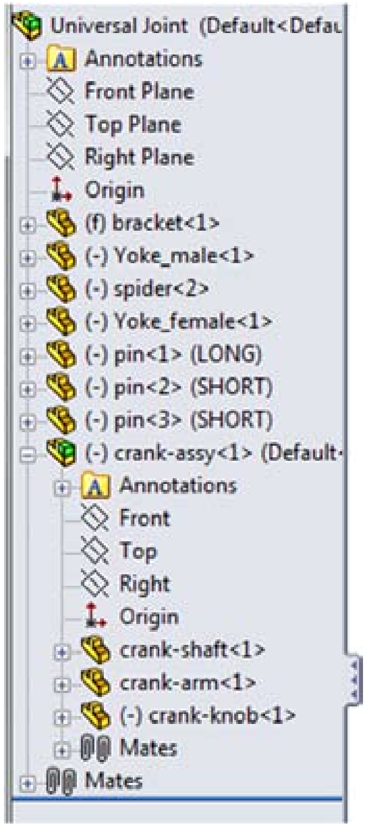


51. Using **Move Component**, click the **Yoke_male** and move it. Orient the flat face on the D-shaped shaft of Yoke_male towards the right to make mating easier in the next step.

52. Save the assembly.

Sub-assemblies

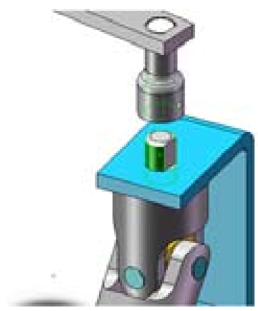
Existing assemblies can also be inserted into the current assembly by dragging. The sub-assembly and all its component parts are added to the FeatureManager design tree. The sub-assembly must be mated to the assembly by one of its component parts. The assembly is treated as a single piece component, regardless of how many components are within it.

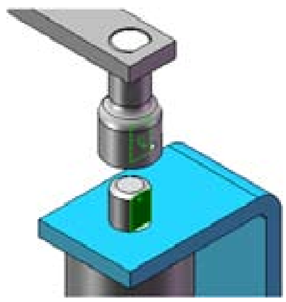


53. Drag the assembly **crank-assy** into the assembly from the **Explorer**.

    The sub-assembly with all its components and mates are added as a component icon. Expanding the sub-assembly component icon shows all the component parts within it, including its own mate group.

54. Add a **Concentric** mating relationship between the cylindrical surfaces on the top of the **male_yoke** and the **crankshaft**.





55. Mate the flat face on the Yoke_male with the flat face in the hole in the crankshaft using a parallel mate.

    Question: *Why wouldn't you use a* ***Coincident*** *mate here?*
    Answer: *Because unless the dimensions of the flats and the diameters of the shaft and corresponding hole are exactly right, a coincident mate would over define the assembly.*

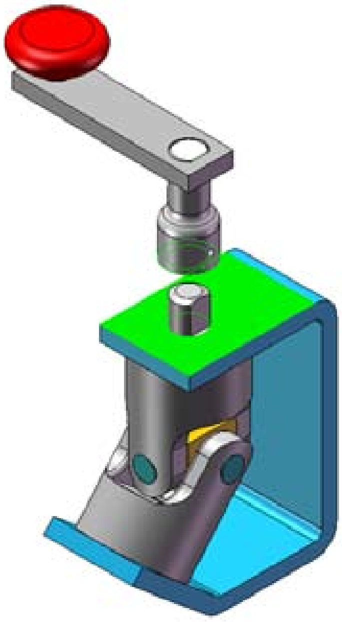
56. Use **Move Component** to rotate the sub-assembly around.

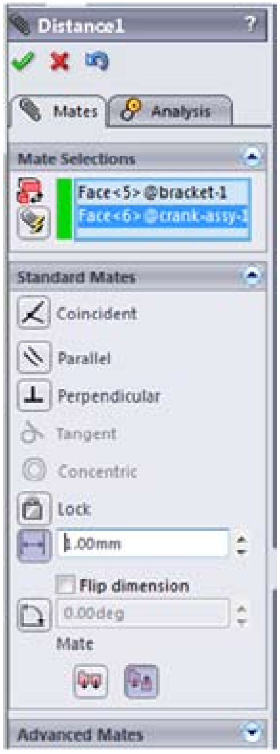
57. Check to see that the flat faces are properly mated.

Distance Mates

Distance mates allow for gaps between mating components. You can think of it as a parallel mate with an offset distance. There is generally more than one solution, so the options **Aligned**, **Anti-Aligned (On)** and **Flip Dimension** are used to determine how the distance is measured and what side it is on.

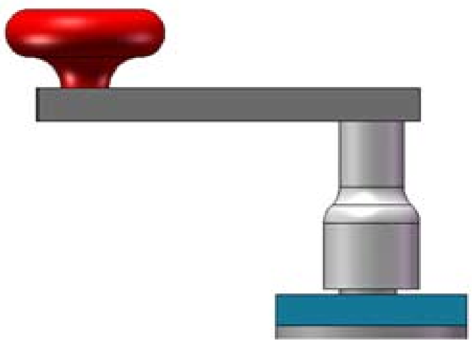
58. Select the top face of the bracket and the bottom face of the crank-shaft component.





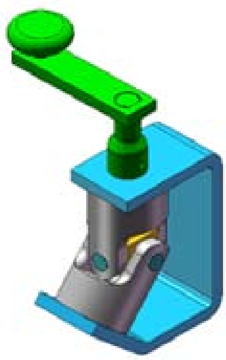
59. Add a **Distance** mate with a distance of **1 mm**.

The **Preview** graphics shows that the distance is measured on the proper side of the mate. If the direction was wrong, the **Flip Dimension** checkbox would reverse it. Press **OK** to create the mate.



60. Select the sub-assembly **crank-assy** in the FeatureManager Design Tree.

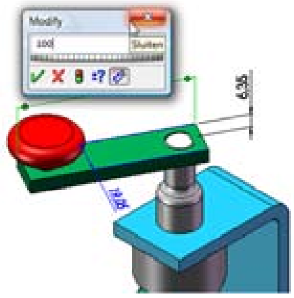
All components in the sub-assembly will be selected and highlighted light green.



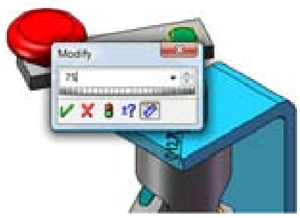
Editing the Assembly

SolidWorks gives you direct access to all the parameters of the components that are in the assembly. Editing them is as simple as double-clicking on a component to display its dimensions and then double-clicking on the dimension you wish to change. The **Annotation** option **Show Feature Dimensions** is also available in assemblies. Click on the **Rebuild** icon and the assembly updates.

61. Double-click on the graphics of the **crank-arm** part to display its dimensions. These are the dimensions used to build the part. Double-click on the **length** to display the Modify window and change the value from **75 mm** to **100 mm**.



62. Double-click on the graphics of the **crank-shaft**. Double-click on the **upper cilinder** and change the value of the length from **40 mm** to **75 mm**.



63. Rebuild the assembly by clicking **Rebuild** .

    Note: *Not only are the parts rebuilt and the assembly updated, the mating relationships ensure that the crank-arm moves up when the crank-shaft gets taller and the knob moves when the crank-arm gets longer.*

    Changing a part at the assembly level changes it at the part level and visa-versa. That is because it is the same part, not a copy.

64. Right-click **crank-shaft** in the FeatureManager design tree and **Open** the component part.

65. Change the value back to **40 mm** and close the part, saving the changes.

    Changes have been made to a reference of the assembly, in this case the size of a part. Upon re-entering the assembly, SolidWorks asks whether you want to rebuild and save the assembly.

66. Click **Yes**.

67. Select and change the dimension of the crank-arm back to **75 mm** and **rebuild**.

An Introduction to Assembly-Centric Design

You may have noticed that there is no hole in the bracket that corresponds to the cut feature in the bottom of the Yoke_female. In a component-centric design environment, trying to position the hole correctly in the bracket would be difficult at best. However, in the assembly-centric environment of SolidWorks, it is simple.

The process of modeling in the context of an assembly can be broken down into a few simple steps:

- Select the part you wish to modify and use the **Edit Part** command.
- Select an appropriate sketch plane and open a sketch as you normally would.
- Reference the needed geometry in the mating part(s).
- Create the feature, in this case a cut.

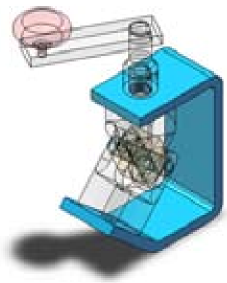
While you are working in an assembly you can switch modes between editing the assembly – adding mate relations, inserting components, etc. – and editing a specific part. Editing a part while in the context of an assembly allows you to take advantage of geometry and dimensions of other components when creating mating features. **Edit Part** can be found:

Select the part you wish to edit and:

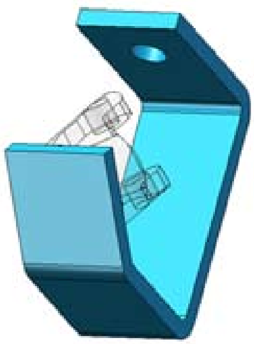
- From the pull-down menu, choose **Edit, Part**
- From the right-click pop-up menu, choose **Edit Part**
- Pick the **Edit Component**  on the CommandManager. This icon is a **toggle**. It lets you switch between **Edit Component** mode and **Edit Assembly** mode.

68. Select the **Bracket** component and switch to **Edit Part** mode.

    When editing a part in the context of the assembly, the active part remains visible while the rest of the assembly turns to transparency mode.

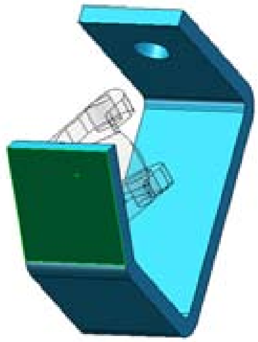


69. Select and hide all the components except the **bracket** and **Yoke_female**. The easiest way to make this selection is to use the FeatureManager design tree. Shift-select the entire list of components and Ctrl-select the two you wish to remain showing to exclude them from the list.

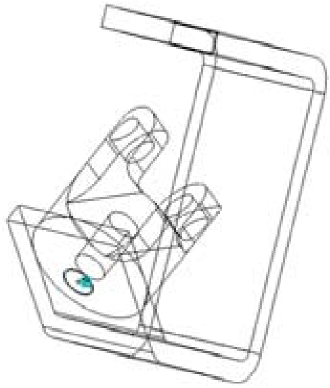


70. Orient the view so you can easily see the underside of the angled leg of the bracket.

71. Select the outside, angled face of the bracket as shown and Open a new **Sketch**.

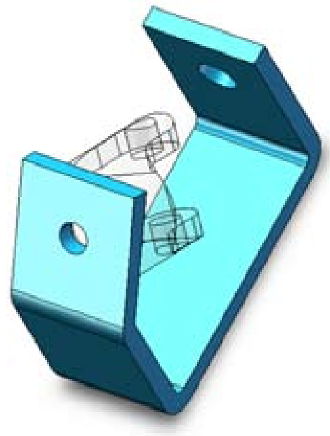


72. Set the **Display Style** to **Wireframe** and select the edge of the hole in the **Yoke_female.**

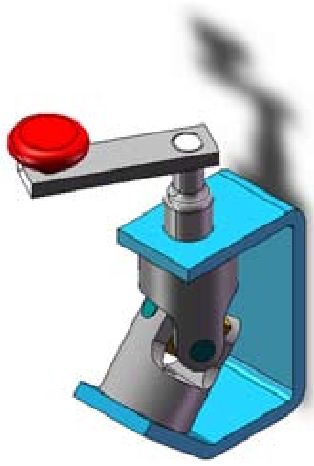


73. Copy this edge into the active sketch using **Convert Entities** .

74. Insert a **Cut** feature in the bracket. Use **Up To Next** and check the preview to ensure the cut is going the correct direction.



75. Switch back to edit assembly mode by right-clicking and choosing **Edit Assembly: Universal Joint**. Alternatively, you can click on the icon to toggle off **Edit Part** mode, returning you to **Edit Assembly** mode.



76. Show all hidden components.

The illustration to the right shows the finished assembly.

In the bracket, the **Cut-Extrude** feature's sketch has an external reference arrow to show that it references geometry outside of the bracket itself.

Component Replacement

You may substitute one part for another in an assembly. If the replacement part has the same face names as the one it is replacing, the mates will be transferred to it. One way to do this is to copy the mated part and make changes to it. For this excercise, a revised version is made of the **Bracket**. You will replace the **Bracket** with this **RevBracket** part.

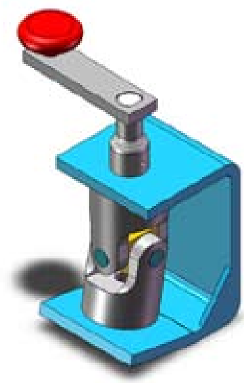
77. Right-click the **Bracket**, and select **Replace** (You might need to expand the menu first by clicking the downward arrows at the bottom). Alternatively, Select **File, Replace...** The **Replace** PropertyManager appears.

78. If not already selected, select in the **Replace these component(s)** box the **Bracket** from the FeatureManager design tree.

79. Click the **Browse** button and double-click the part **RevBracket** from the same **/U-Joint/** directory as **bracket**.

80. Click **OK**.

81. A second menu appears, in which you can adjust the copied mates. In this case they are correct, so click **OK** again to replace the part.

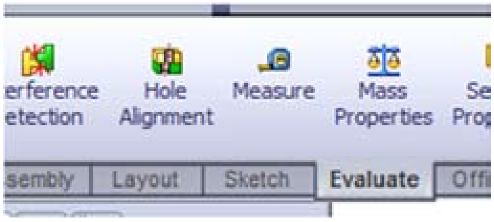


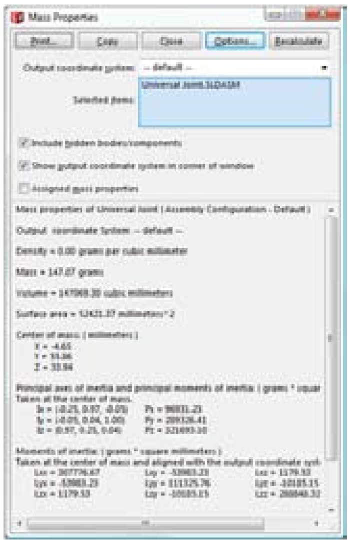
The part RevBracket has replaced the Bracket in the assembly. The mates used with the bracket have been transferred to the RevBracket. If any mates had failed, they could be edited and redefined using the appropriate faces in the replacement part.

Analyzing the Assembly

There are several types of analysis you can perform on an assembly. These include calculating the mass properties of the assembly and checking for interference. One of the types of assembly-based analysis you can perform is the calculation of the mass properties of the assembly.

82. Click **Mass Properties** from the **Evaluate** tab, or **Tools, Mass Properties**.





The system performs the calculations and displays the results in a report window. The system also displays the **Principal Axes** as temporary graphics. **Options** can be used to change the units of the calculation.

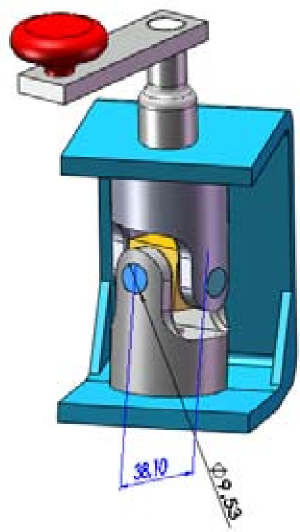
83. Click **Close**.

Detecting and Correcting Interference

An important aspect of designing an assembly is making sure all the parts fit. There are two possible ways to check this: **Interference Detection** (static) and **Collision Detection** (dynamic). We will use both in an example.

Interference Detection takes a list of components and finds interferences between them. The interferences are listed by paired components including a graphic dimensioned box representing the interference.

Since the current model is correctly modelled, there is no interference. For the sake of this example, we will create an error ourselves.



84. Double-click on the **LONG** version of the **pin** in the graphics area to show its dimensions.

Tip: If you don't know which one the LONG pin is, select it in the FeatureManager Design Tree to highlight it.

85. Change the diameter from **9,53** to **10 mm**.

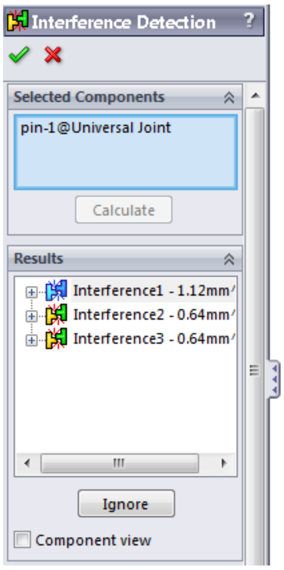
86. Rebuild the assembly by clicking **Rebuild** .

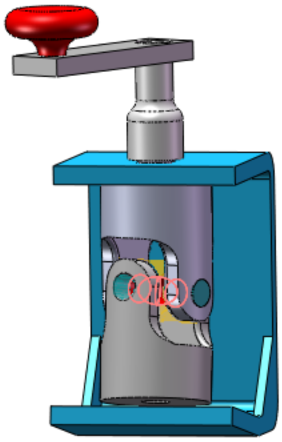
The **pin** is now slightly larger, making it too big for the holes of the **Spider** and the **Yokes**.

87. Select the **LONG pin** again, and click **Interference Detection** From the **Evaluate** tab. The **Interference Detection** PropertyManager appears.

88. Make sure only the **pin** is selected under **Selected Components**.

89. Click the **Calculate** button.





Under **Results**, all the interferences are shown. One interference with the **Spider** and two with the **Female_Yoke**, as predicted. By selecting an interference, it will be marked in the graphics area as as a red colored face. If you expand the interference by clicking the plus sign + , the interfering components are shown.

90. Click **OK** to close the **Interference Detection**.

Note: instead of one component, you can also select the whole assembly to view all interferences.

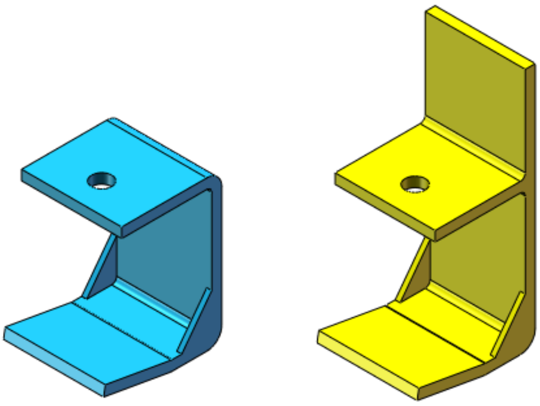
Once a interference is detected, the next step is to correct it. In our case this means changing back the diameter of the pin.

91. Change the Diameter of the **LONG pin** back to **9,53 mm**. if you check the **Interference Detection** again, you will see there is no interference.

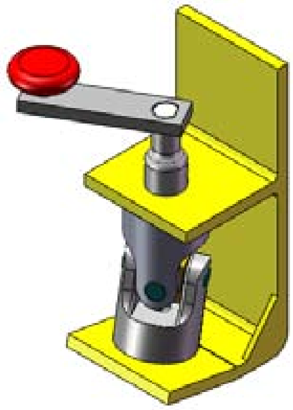
The problem with a static method of interference detection is that the components of the assembly sometimes only interfere under certain conditions. This can be the case with moving parts. This brings us to the second method to check interference: **Collision Detection**.

Collision Detection analyzes selected components in the assembly during dynamic assembly motion, alerting you when faces clash or collide. This is used to remove interferences, or check an intended limited movement. You have the opti-ons of stopping the motion upon collision, highlighting the colliding faces, and generating a system sound.

But again, before continuing, we will change the assembly to suit it for this example. We will replace the **RevBracket** for the **CollisionBracket** of the same directory. This **CollisionBracket** has an extra wall on the back, intended for limiting the movement of the **Crank**.



92. Right-click the **Bracket**, and select **Replace** , or select **File, Replace...** The **Replace** PropertyManager appears.



93. Browse for the **CollisionBracket** part in the **/U-Joint/** directory**.**

94. Click **OK** twice.

95. Use **Move Component** to turn the crank in a collision-free position, if necessary. Don't exit the PropertyManager yet.

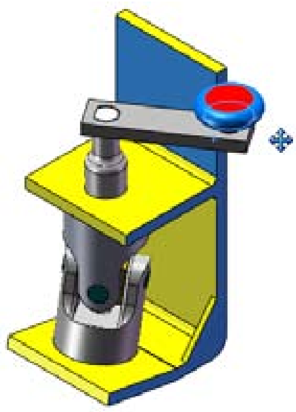
96. Select **Collision Detection** in The **Options** box to enable dynamic collision detection.

97. Check the **Stop at collision** box.

The option **All components** means collisions with all assembly components are detected. This puts more demand on system resources, especially in a large assembly. If you choose **These components**, only collisions with a group of assembly components that you select are detected.

98. Expand the **Advanced Options** panel. Make sure **Highlight faces** and **Sound** are checked in the **options** box.

99. Turn the **U-joint** by dragging the handle.



When the crank and the bracket collide, the system alerts you by highlighting the faces.

100. Turn off the **Move Component** tool.

101. Replace the **CollisionBracket** with the old **RevBracket** again.

Exploded Assemblies

You can make exploded views of assemblies automatically or by exploding the assembly component by component. The assembly can then be toggled between normal and exploded view states. Once created, the exploded view can be edited and also used within a drawing.

Enter the exploded view dialog to create the view. The dialog offers different methods of creating and editing the exploded view. The **Explode Steps** option will be used here, which means that exploding the assembly will take place one part at a time. Exploding a part can be found:

- From the menu pick **Insert, Exploded View,**
- Or from the CommandManager click **Exploded View** .

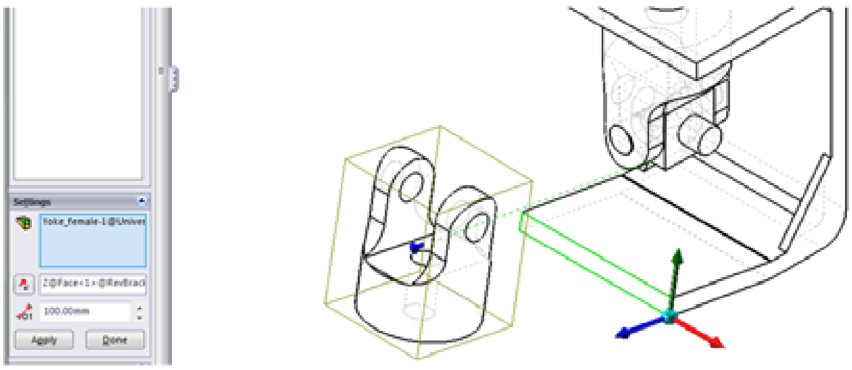
102. Click **Exploded View** . The **Exploder** PropertyManager appears.

The first component to be exploded will be the **Yoke_female**. It will be exploded downward normal to the angled face of the RevBracket.

Note: *The graphics that follow will be in **wireframe mode** for a clearer view of the direction arrows and selections.*

103. Select the component **Yoke_female** from the screen or the FeatureManager design tree. The component name will appear in the **Components of the explode step** list.

104. Select the front **angled face** of the **RevBracket** component as the **explode direction**.



Note: *When you select a face a triad appears. Click the arrow of the direction you prefer.*

105. Set the **Explode Distance** to **100 mm**.

106. Click **Apply** for a preview. The component moves the 100 mm in the direction indicated.

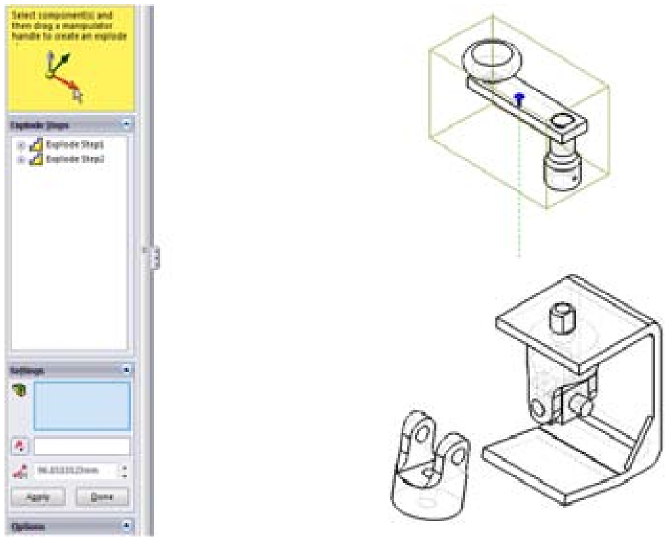
107. Click **Reverse Direction** for changing direction if needed.

108. Click **Done** for a new step. **Explode Step1** is now added to the **Explode Steps**. Do **not** press **OK**.

Entire sub-assemblies can be exploded as a single component or as individual ones. Check **Select sub-assembly's parts** when you want to explode sub-assemblies. You can also move parts by **Selecting** and **Dragging** the **direction arrow** of the triad. If an **Explode Direction** is not selected, the **Universal XYZ** of the **assembly** will be used.

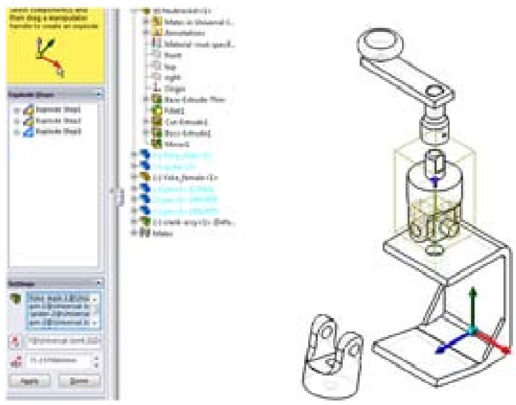
109.Add a new explode step. Choose the sub-assembly **crank-assy** as the **component** and the **top face** of the **RevBracket** as the **direction**. The assembly must be chosen from the FeatureManager design tree to select all of the components in it. Make sure that the **Select sub-assembly's parts** option is **not** checked to move all the sub-assembly components as one.

110. Drag the sub-assembly upwards roughly as shown below, using the **vertical arrow**. the step is automatically completed.



Multiple selected components can be exploded as a single group if the components are selected together or if they are related. Multiple selections are allowed in the **Components of the explode step** list.

111. Select **Yoke_male**, **Spider** and the three **pins** for **Components** and **drag** them **upwards** into position between the **RevBracket** and **crank-assy** as shown below. **Explode Step3** is complete.

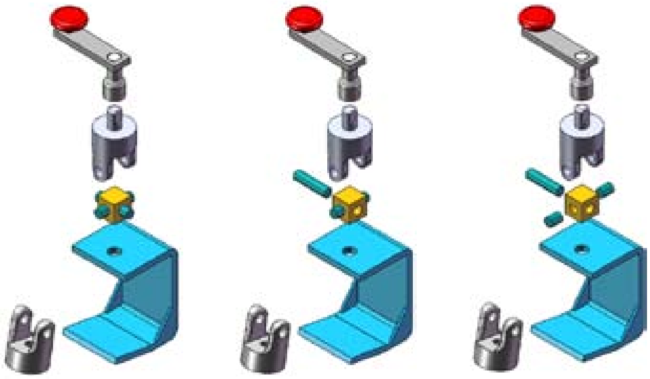


Any existing explode step can be recalled and edited. By selecting it, the relevant entities are highlighted and you can change the distance. **Double-clicking** the step opens it again, allowing you to fully change it. Alternatively, **right-click** the step and select **Edit Step** opens it as wel.

112. In the **Explode Steps** list box, choose the **Explode step** of the **sub-assembly**.

113. A small **blue drag handle** appears. **Drag** it to reposition the **crank-assy** further upwards.

Subsequent Explosion steps are also possible. We will use this to explode the combined component block of **step 111**.



114. Select the **Yoke_Male** (**not** its Explode Step). Drag it **upwards** to position it above the **Spider**.

115. Select the **LONG pin**. **Drag** to the **left** of the **Spider**.

116. Move both **SHORT pins** to the **front** and **back** of the **Spider**.

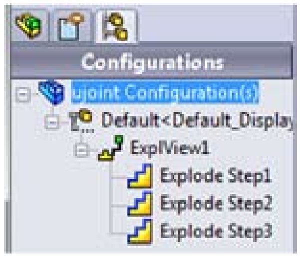
117. Click **OK** to close the Exploder PropertyManager.

118. Save the assembly.

119. To the left, right-click the **ExplView** and choose **Collapse** to return the view to its normal state.

Exploded views are related to and stored in what are called **Configurations**. You can only have one exploded view per configuration. For right now, we need to explain just a few things because they relate to exploded views.

Configurations are listed and managed from within the same window that is occupied by the FeatureManager design tree.



To switch the display of this window, use the tabs located at the top of the window pane. Clicking the  icon will display the Configuration Manager with the default configuration listed. The default configuration is named Default. This configuration represents the part as you modeled it. When you want to switch back to the FeatureManager display, click the  icon.

Once an exploded view has been created, it can be edited in several ways.

120. if not already, switch to the **ConfigurationManager**  tab again.

121. Expand the listing of the **Default** configuration and double-click on **ExplView** to return to the exploded view display. **ExplView** also expands its tree, showing its explode steps.

With a **right-mouse** click on **ExplView**, you can:
- **Collapse** the assembly,
- **Explode** the assembly,
- **Edit** the feature and its explode steps,
- **Animate** the explode of the assembly

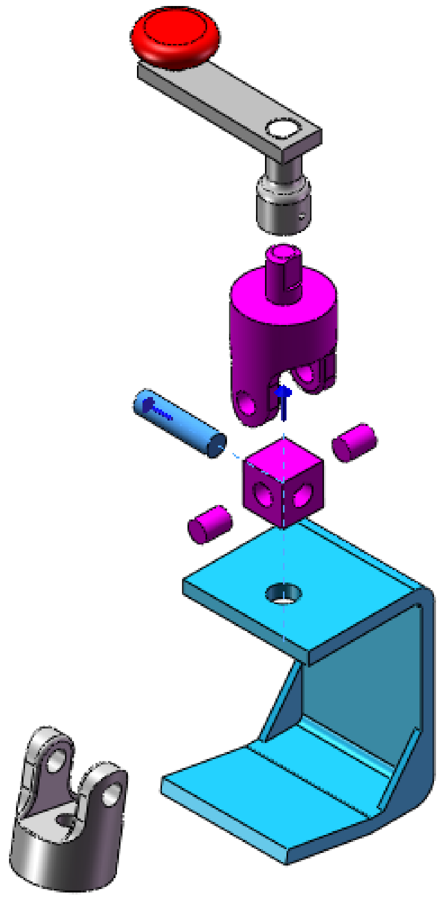
Tip: For small modifications, you can also select an explode step directly from the ConfigurationManager design tree. The entities of the explode step will highlight and the blue drag handle appears again, allowing you to modify it.

122. Switch back to the FeatureManager design tree using the icon.

The view is still in exploded state but now the components are visible.

123. Right-click the **LONG pin** in the FeatureManager design tree and choose **Show Explode Steps**. All the steps used to explode that component are shown as drag handles.

124. save your work. Collapse the exploded view.



Display States

Next to Exploded Views, you can also create different Display States. Display States are used to, among others, save different configurations concerning the appearance. Examples are changing the colors to clarify different parts of the same material, or changing the transparancy, to view difficult assemblies without having to use a Section View.

125. Right-click in the ConfigurationManager, and select **Add Display State**. A new Display State appears in the **Display States** box.

126. Right-click the new Display State and select **Properties**. Change the name to '**Transparent View**' and click **OK**.

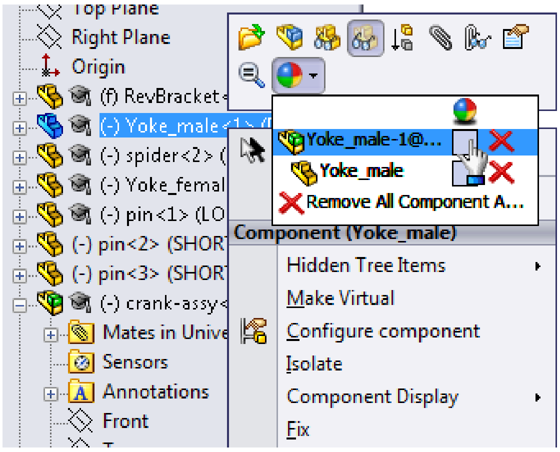
127. Go back to the FeatureManager. Right-click **Yoke_male** and select **Change Transparency**. Repeat this step for **Spider**.

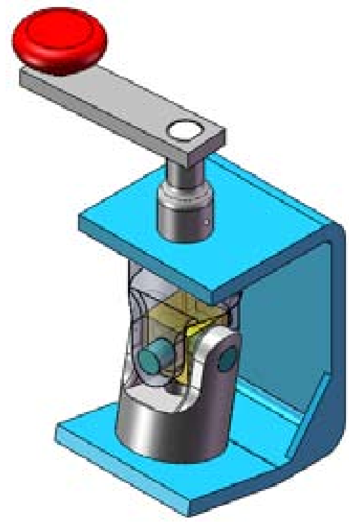
SolidWorks adds a new appearance on **Assembly level**, with the same preferences as the original appearance, except for the transparency. The default rate for transparency is **0.75**.

128. Right-click **Yoke_male** again, select **Appearances** and click **Edit Color** on **Assembly** level, as shown to the right.

129. Under **Optical Properties**, set the **Transparency** to **0.50**.

130. Under **Display States**, select **Specify display state**, and select **Transparent View**. Click **OK**.





131. Right-click the **Spider**, select **Appearnces** and click **Edit Color** on **Assembly** level.

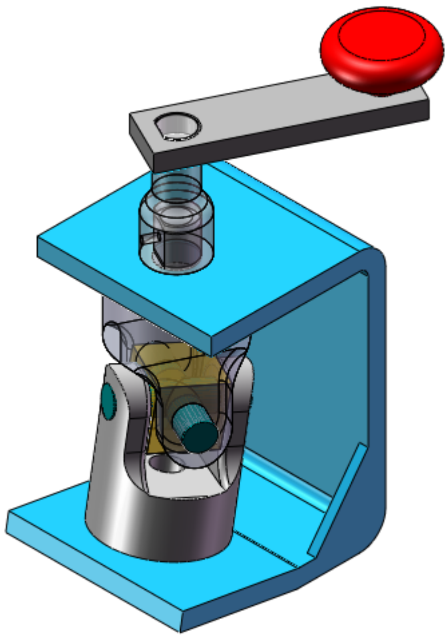
132. Set the **Transparency** to 0.30 and link this Appearance again to the **Transparent View** Display State. Click **OK**. The part should now look like the example to the right.

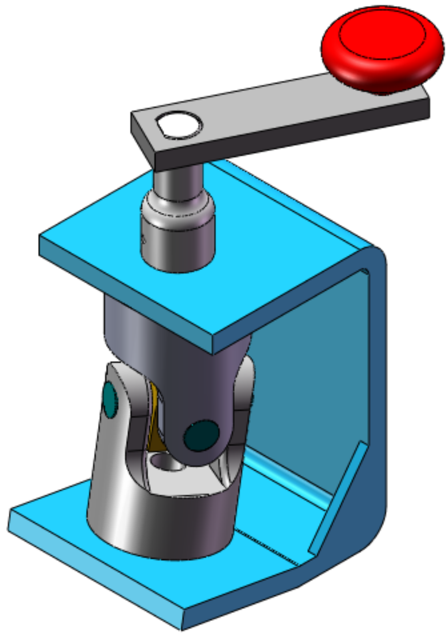
133. In the design tree, expand the '**crank-assy**' assembly. Right-click '**crank-shaft**' to **Change** its **Transparency**.

134. Edit its color the same way you did with the first two parts. Set the **Transparency** to **0.50**, link it to the **Transparent View** Display State and click **OK**.

135. Save the assembly. If it is asked to rebuild, click **Yes**.

136. Turn the crank and look at the new display state. Go to the **ConfigurationsManager** tab and double click the default view.





Using Physical Dynamics

Physical Dynamics is a method for visualizing assembly motion in a more realistic way. Expanding on the capabilities of dynamic collision detection, **Physical Dynamics** lets objects act upon another. When two objects collide, one will move the other according to the available degrees of freedom. **Physical Dynamics** propagates throughout the assembly. The dragged component can push aside a component, which then moves into and pushes aside another component, and so on.

Note: *Do not confuse* ***Physical Dynamics*** *with a kinetic analysis application. With* ***Physical Dynamics****, characteristics such as momentum, friction or whether a collision is elastic or inelastic are not considered.*

Physical Dynamics can be found:

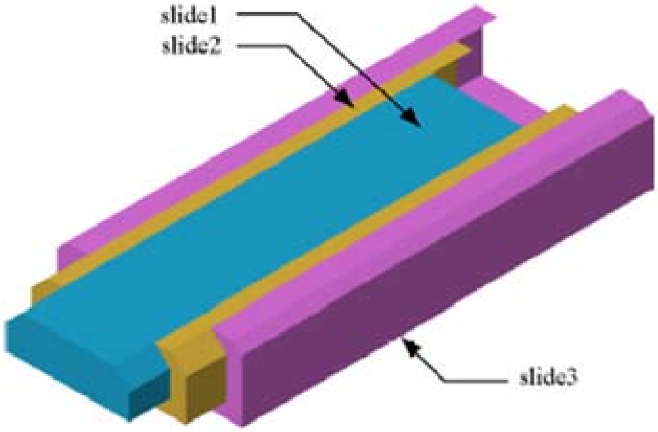
• On the Move Component PropertyManager, click **Physical Dynamics**.

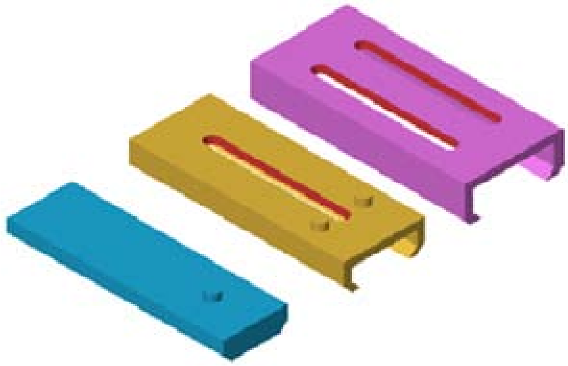
When you drag a component with **Physical Dynamics** enabled, a small symbol appears on the component. This represents the center of mass. **Physical Dynamics** uses mass properties to compute in which way the forces acting on a component will make it behave as it collides with other components. Dragging a component by its center of mass exhibits different motion than dragging by a point on the component.

1. Open the existing assembly **'Nested Slides'** from from the **'Chapter 3/Nested Slides'** directory.

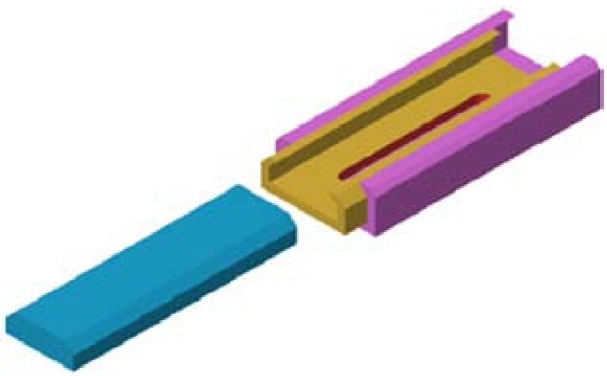
The assembly contains three component parts. They are **slide1**, **slide2**, and **slide3**, as shown below. The slides have pins and slots. In an actual product the slots would limit the distance the slides can travel.

Note: *The assembly is shown exploded for illustration purposes.*

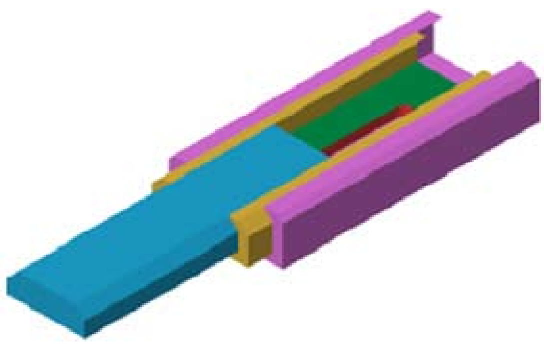




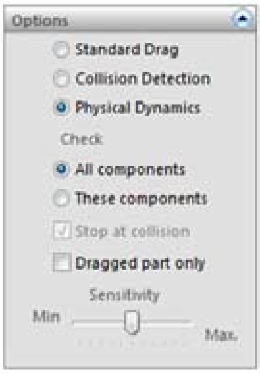
The slide3 is fully constrained. The two inner slides each have one degree of freedom. If you drag slide1, it will slide completely out of the assembly.



If you turn on Collision Detection and Stop at Collision, the slide1 will stop when the pin reaches the end of the slot in slide2, but it will not drag slide2 along with it. So, this still does not behave like the actual physical model would.

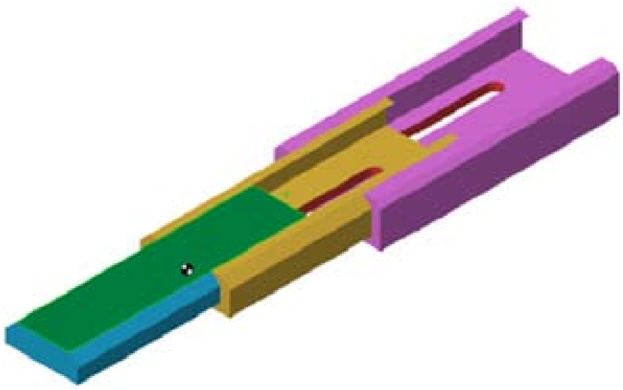


2. On the **Move Component** PropertyManager, click the **Physical Dynamics** option located in the **Collision Detection** group box.



3. Drag **slide1**. When its pin reaches the end of the slot in slide2, it drags it too. This continues until the pins in slide2 reach the end of the slot in slide3.

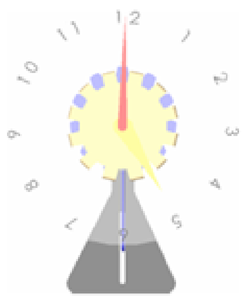
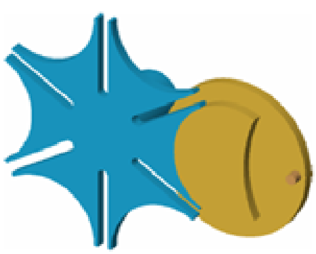
   This is a much better representation of how the actual physical model would behave.

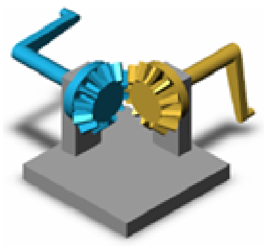
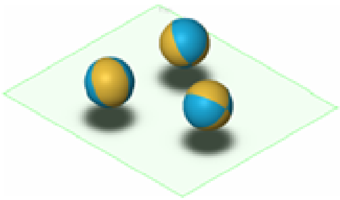
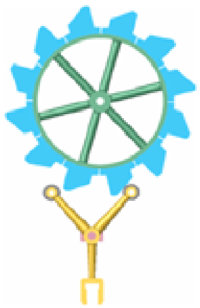


4. Exit the assembly, saving your work in */work directory/* **Nested Slides**.

Other examples

In the Physical Dynamics folder are some other examples. They are illustrated in the chart below and are also available in the ‘**Chapter 3**’ directory.

Assembly	Behavior
Clock 	As you drag the minute hand, the hour hand moves.
Geneva Wheel 	As you turn the input wheel, the pin engages and disengages the slots in the output wheel.

Assembly	Behavior
Bevel Gears 	Turn the handle on one gear, and the other gear rotates.
Rolling Balls 	Drag the individual balls so they collide wit each other.
Limit Mechanism 	Rotate the cam wheel counterclockwise and the Y-shaped actuating lever oscillates back and forth.

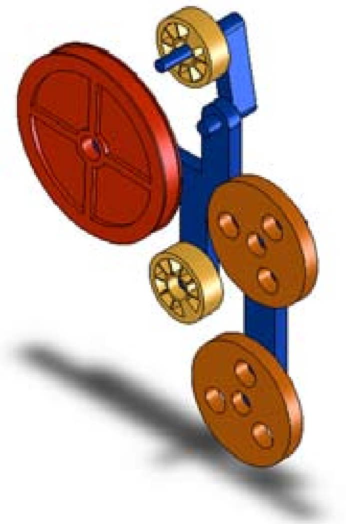
Tips for Working with Physical Dynamics

There are some things you should keep in mind when you use **Physical Dynamics**.:

- **Physical Dynamics** depends on collision detection. It will not work if the assembly contains interferences. If the item you are dragging interferes with another component, the source of the interference is made transparent. Use **Tools, Interference Detection** to find and eliminate interferences before using **Physical Dynamics**.
- Use the appropriate mates to define the assembly. Highly unconstrained assemblies are less likely to be successful. Do not depend on **Physical Dynamics** to solve everything. For example, in the Nested Slides assembly, the appropriate mates were used to mate slide1 and slide2 so they each had only one degree of freedom. Then **Physical Dynamics** was used to handle the interaction of the pins and the slots.
- **Physical Dynamics** does not work on assemblies that have symmetry mates.
- **Physical Dynamics** can be computationally intensive. Limit the scope by selecting components in the Selected Items box, and then click **Resume Drag**. Items that are not in the list are ignored.

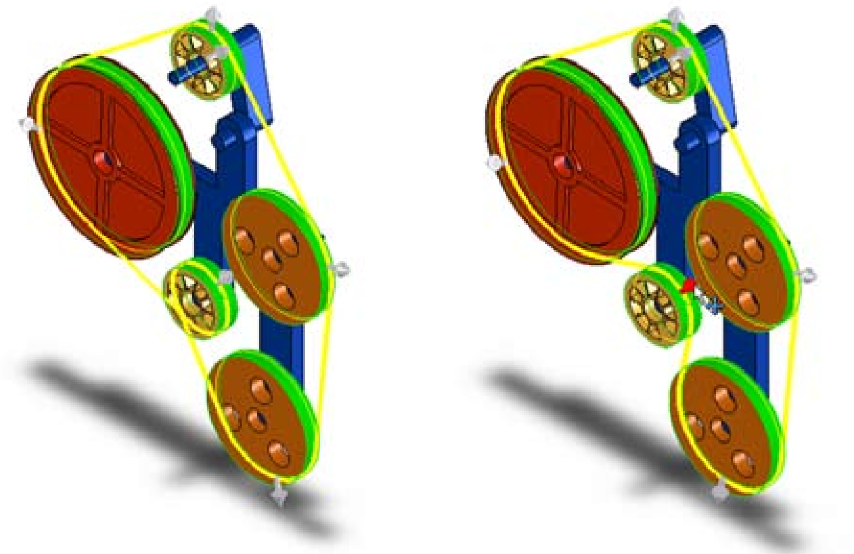
Belt/Chain

*Note that this paragraph is **Optional**.*



With the help of belt and chains it is possible to create a relation between disks and other circular parts. The belt (or chain) transfers the angle of rotation from one disk, to another disk.

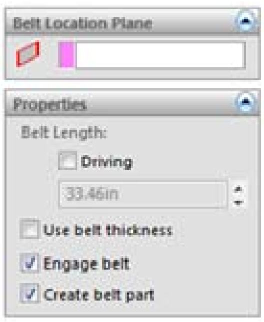
1. Open **pulleys.sldasm** from the **pulleys** folder.
2. Select the **isometric view** of the assembly.
3. Click **Insert, Assembly Feature, Belt/Chain**.
4. Select all **cylindrical faces** of the 5 disks by clicking on them, **clockwise**. If the order of disks is not right, use the two arrows of the **Feature Manager** to set the disks in the right order.



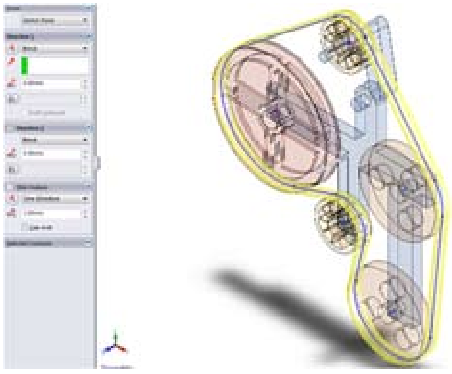
5. Change the rotate direction of the belt from the lower small disk, by clicking on the **direction arrow**, or press **Flip Belt Side** in the PropertyManager. The belt now runs above the small disk.

6. Set the diameter of the small lower disk to **46 mm**.

7. Select **Engage belt** and **Create belt part** from the **Properties** tab.



8. Click **OK**. To **save** the **belt part**, save the assembly. A message box appears to **Save all modified documents** within the assembly. the part. Choose **Save All**. In the next **Save As** menu, choose **Save Externally**, name the part **Belt.SLDPRT** and Specify the **Pulleys** folder as **Path**, if nog already selected. click on **OK**.



9. Expand the **Belt** feature in the FeatureManager design tree. Select the **Belt<1>** part and click **Edit Component** .

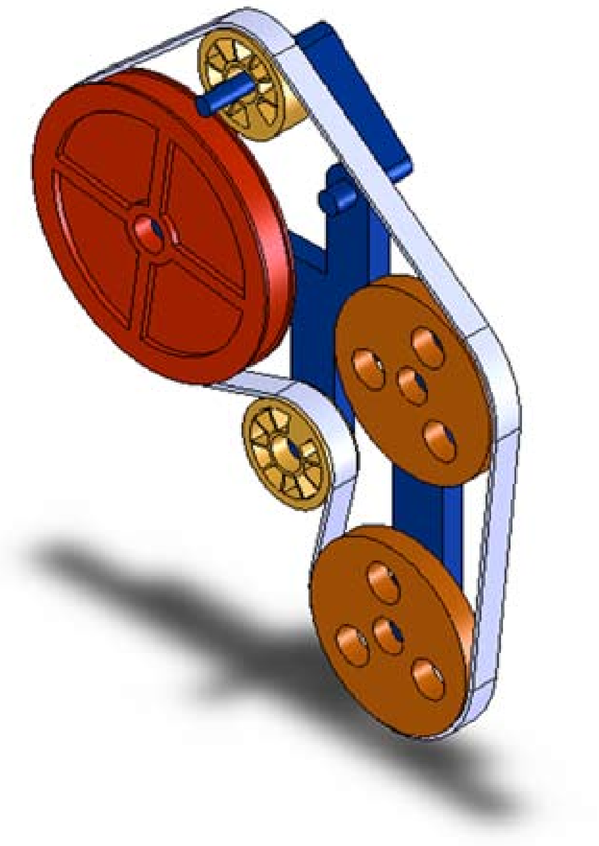
10. Expand **Belt<1>** as wel and select the **Sketch**. Make an Extrude by clicking on **Extrude Boss/Base** .

11. check the **Thin Feature** box and set the thickness to **2 mm**. Set the **end condition** to **Mid Plane** and **depth** to **6 mm**.

12. Click **OK** and deselect **Edit Component** from the assembly menu.

13. Check the working of the belt by rotating one of the disks. If it works correctly, all disks rotate with the same speed.

14. Save the part.



Chapter 4: Drawings

This chapter describes what a drawing is and how you can create one. By means of examples and exercises it is explained how and which features can be used.

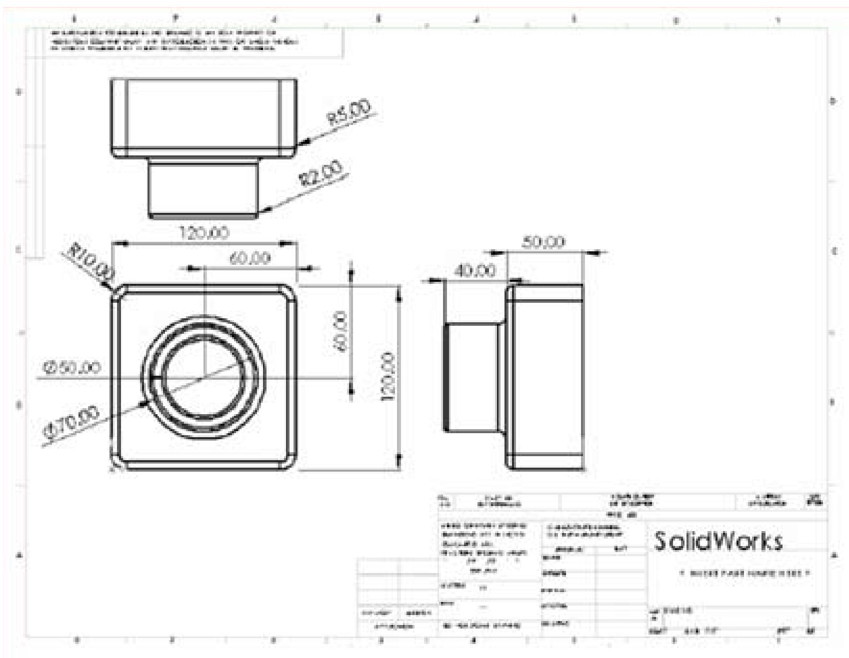
Drawing Basics

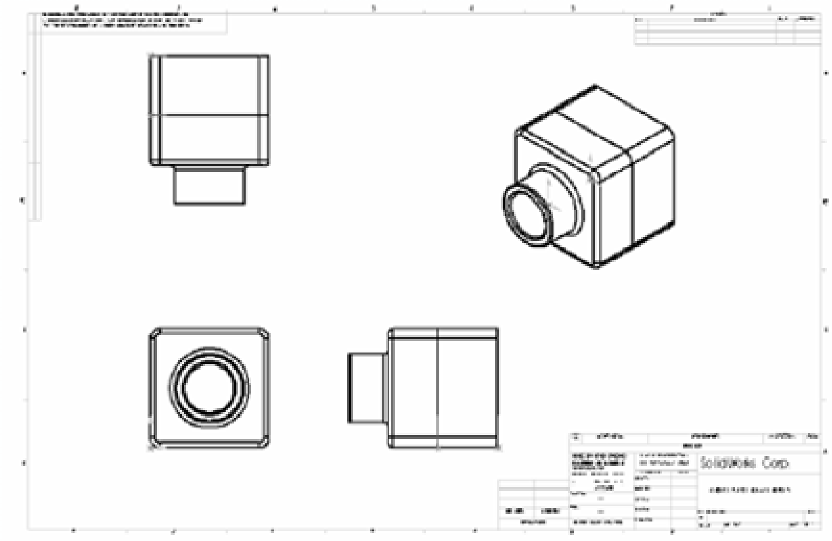
A drawing is a two-dimensional representation of a model.

You can create 2D drawings of the 3D parts and assemblies you have learned to create in the previous chapters. Parts, assemblies and drawings are related to each other, every change that is made in a part or assembly document will be implemented in the drawing document.

A drawing usually consists mostly of one of more views of a model, and sections of the model to show the internal structure.

The first paragraph of chapter 4 will teach how to create the drawing below and the one on the next page. It concentrates on the creation of a drawing, drawing views, and the addition of model dimensions to the drawing.





Some of the important topics in the lesson are listed below. Each of these topics comprises a section in the lesson.

- **New Drawing**
  Drawing files can have multiple sheets, each with multiple views. The views can reference different parts or assemblies.
- **Drawing templates**
  SolidWorks provides standard templates for drawing sizes both ANSI and ISO, as well as the ability to create your own. A template contains the correct sheet format as well as predefined fonts, arrows, projection method, etc.
- **Drawing views**
  Drawing views can be created in many ways including standard, named, section, detail, and aligned section for parts and assemblies.
- **Model dimensions**
  The dimensions in sketches and other features that were used to create the model and be imported into the drawing view directly. This type of dimension is considered "driven", and can be used to make changes in the model from the drawing sheet.

Drawing Templates and Sheet Format

Before creating drawings, we must first install the drawing templates used by the University of Twente.

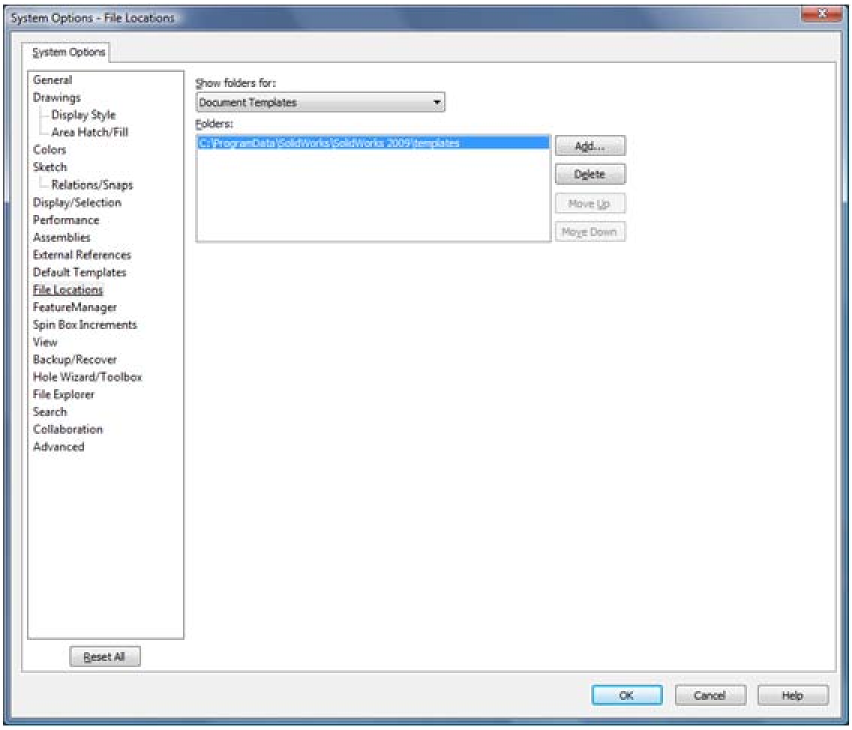
1. If you haven't done so already, **download** the drawing templates, **SW2010_UT-templates.zip**. They can be found on the course site of Blackboard or http://www.opm.ctw.utwente.nl/staff/onderwijs/cadcam/solidworks.htm .

2. Unpack **SW2010_UT-templates.zip** to a convenient location.

*Note: the steps here are similar to the ones in the powerpoint presentation, **Installeren UT templates.ppt**, located within the zip archive. The presentation is more illustrated.*

Within the zip archive are three types of templates that have to be installed, **Drawing Templates**, **Sheet Format Templates** and a **Bill of Materials Template**. we will install these one by one.

3. Go to **Tools, Options** and click on **'File Locations'**.

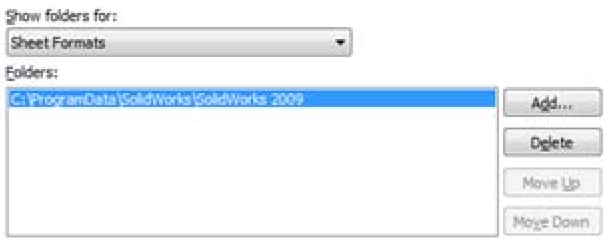


4. Select **Document Templates** from the pull down menu, if not already selected.

all the files from the **drawing-templates** folder need to be copied to the folder shown above, highlighted in blue. This location varies between different SolidWorks versions and Operating Systems, so it is important that you use the folder path shown in your own Options menu (and not the path shown in the images of this manual).

5. **Copy** the files from the **drawing-templates** folder to the right folder shown in your Options menu.

Tip: the Folder Path on your system will probably refer to **C:\ProgramData**. *This is normally a* ***hidden*** *map. To make it* ***visible, Open Windows Explorer.*** *Press* ***Alt*** *to show the* ***menu bar.*** *Select* ***Tools, Folder Options.*** *Go to the* ***View*** *tab. Under* ***Hidden files and folders,*** *choose* ***Show hidden files and folders.*** *Press* ***Apply.***



6. Next, select **Sheet Formats** from the pull down menu.

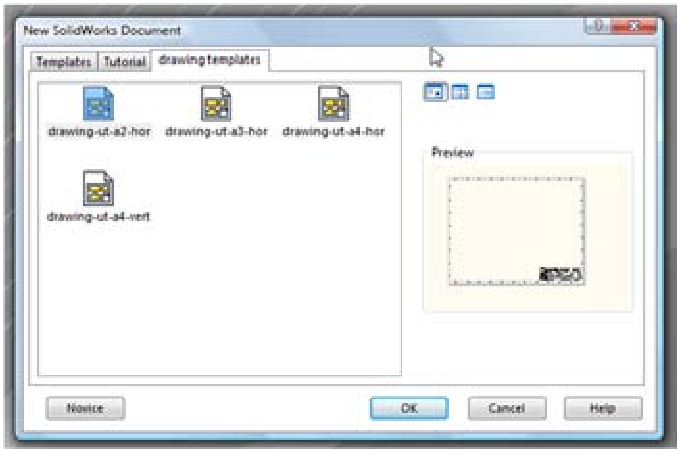
7. **Copy** the files from the **sheet-format** folder to the right folder shown in your Options menu.

8. Lastly, select **BOM Templates** from the pull down menu.

9. **Copy stuklijst_UT** to the right folder shown in your Options menu.

10. **Close** the **Options** menu.

11. Click **New** on the Standard toolbar. The New Document dialog appears.



12. Click **Advanced** at the left bottom of the dialog, to display more options.

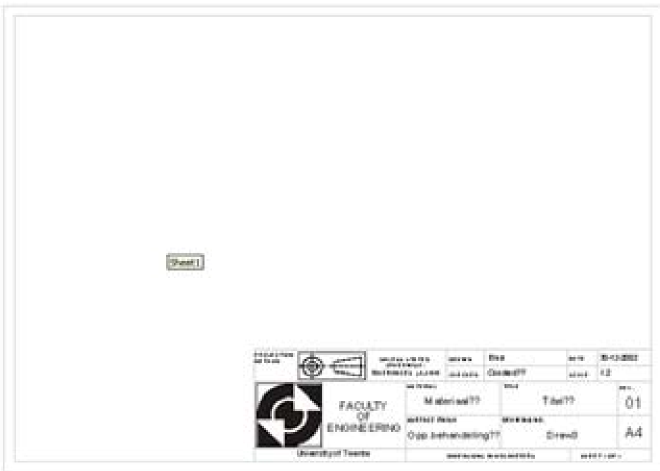
13. Select the **Templates** tab. A window with installed templates appears.

14. Select **drawing-UT-a4-hor** and click **OK**. The **Model View** PropertyManager is displayed. We will add the model later, so **close** it for now.

Once the drawing sheet has been chosen, the drawing is created. The drawing environment includes the graphics area that displays the **Drawing Sheet** and the **FeatureManager**.

- **Drawing Sheet**
  The Drawing Sheet contains a set of views of a part or assembly. These views are oriented, aligned and scaled as you desire. All the annotations that are added to the views are within the drawing. One drawing file can have multiple drawing sheets within it; each sheet having its own set of views.
- **FeatureManager Design Tree**
  The FeatureManager design tree has a somewhat different appearance for drawings. It contains the drawing sheets, views, annotations and references used by the drawing. Views and drawings can be manipulated and changed directly from the FeatureManager menu options. New views and sheets will appear in the FeatureManager as they are created. If you expand the listing for a particular view, you will see the part or assembly, complete with its features that it references.
- **View Orientation**
  Like the dialog in parts and assemblies, View Orientation is used to manipulate the display of the graphics.

15. Zoom in on the title block with **Zoom to Area** .



As you can see, the title block needs some information to be complete.

16. Click **File, Properties** and select the **Custom** tab.

17. Select under **Property Name** ‘Drawn by’. the value ‘Getekend??’ .

18. Change the value to your name.

19. Fill in the other properties and click **OK**.

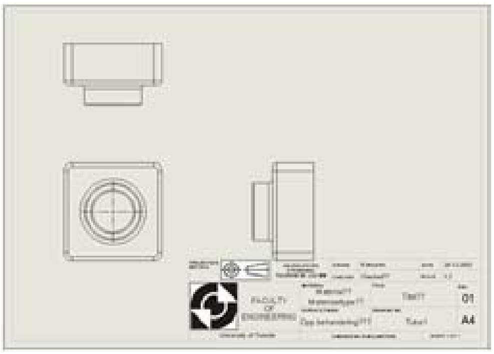
Creating a Drawing of a Part

The core of creating drawing views concerns the creation of **Model Views, Standard 3 Views, Named Views** and **Section Views.** These views are often the initial views in a drawing, from which other views are generated.

20. **Open TUTOR1.sldprt** from your */work directory/.*

21. Return to the drawing and click **Standard 3 View** in the View Layout tab, or **Insert, Drawing View, Standard 3 View.**

    Note: *The cursor shape changes to . TUTOR1 is selected in the* ***Part/Assembly to Insert*** *box.*



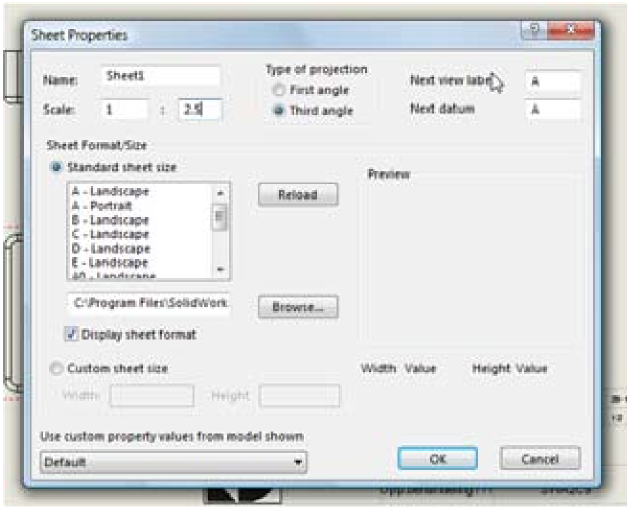
22. Click **OK** to insert a **Standard 3 View** of TUTOR1.

23. Click **Zoom to Fit** if necessary to see the entire sheet.

    Note: *A different way to create a view is to use* ***Insert, Drawing View, Model..*** *In the PropertyManager you can select the part or assembly from which to create the view and then you can select the view you prefer.*

Changing the Scale

When you create a view on a drawing, it is defined at the default view scale. That default scale is controlled by the drawing template. (If a drawing has no template, the default scale is controlled by the settings in **Tools, Options**.)

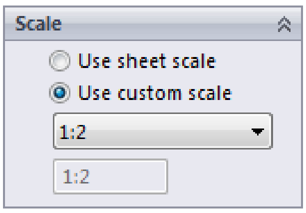


24. Open the **Sheet Setup** dialog by right-clicking **Sheet1** and selecting **Properties** in the FeatureManager design tree.

    Here you can change the default scale, Type of projection (Third angle = American projection) as well as other characteristics of the drawing.

25. Check if the **Scale** is set to **1:2** and click **OK**.

    Changing the **Scale** while in **Edit Sheet Format** mode and saving the template means that all new drawings that use this template will have this view scale by default. To change the scale of a single drawing sheet, edit the properties while in **Edit Sheet** mode.



26. Click within the boundaery of the **Front** view (inside the dashed lines) and check the **Use custom scale** checkbox in the PropertyManager.

27. Change the scale to **1:5** and notice the effect.

28. Change the scale back to **Use sheet scale**.

29. Click on the borders of the two other views and make sure **Use parent scale** is selected.

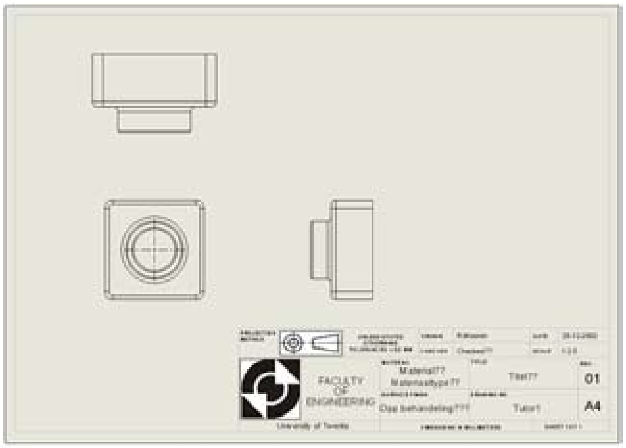
Positioning of views

Drawing views can be repositioned by dragging them around the drawing. In the standard 3 view arrangement, the front view is the source view. This means that moving the front view moves all three views. The top and right views are aligned to the front. They can only move along their axis of alignment.

To move a view, click inside its boundary (the dash lined box), then drag it by its green border.

The top and right views are aligned to the front view, and only move in one direction to preserve the alignment.

- To move the top view vertically, drag up and down.
- To move the right-hand view horizontally, drag sideways.
- To move all the views together, click the front view and drag in any direction.



30. Move the views on the drawing sheet as shown to the right.

31. Save your drawing as **TUTOR1** in your */work directory/*. It gets the extension **SLDDRW**.

Adding Dimensions to a Drawing

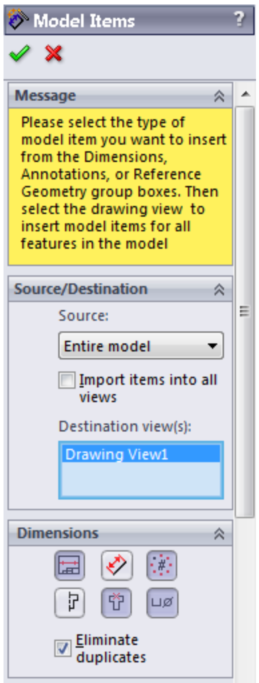
Model dimensions are simply dimensions and parameters that were used to create the part and that have been inserted into the drawing. These dimensions are considered to be driving dimensions. Driving dimensions can be used to make changes to the model. You can insert model dimensions into the drawing in four ways. You can automatically insert all the dimensions associated with the:

- **Selected view**
- **Selected feature(s)**
- **Selected component in an assembly**
- **All views**

Inserting all Model Dimensions

The dimensions created in the part will be used in the detail drawing. In this case all the dimensions in all views will be inserted. When the system inserts model dimensions into all views, it starts with the detail and section views first. Then it adds any remaining dimensions to remaining views based on which views are most appropriate for the features being dimensioned.

Model Items allows you to take the dimensions that were created while modeling and insert them into the drawing. Select a view, feature, or component and choose from the dialog box to add the dimensions you want. It can be found on the **Annotations** tab, or in the pull down menu: **Insert, Model Items...**



32. Click a blank area inside the view boundary of the front view in the drawing, and click **Model Items** .

    The **Insert Model Items** PropertyManager box appears. You can select which types of dimensions, annotations, and reference geometry to import from the model to the selected view.

33. Select **Entire Model** as **Source** and make sure that the **Import Items into All Views** option is **unchecked**.

34. In the **Dimensions** box, select the types as shown to the right. Under **Annotations** and **Reference Geometry**, make sure none are selected.

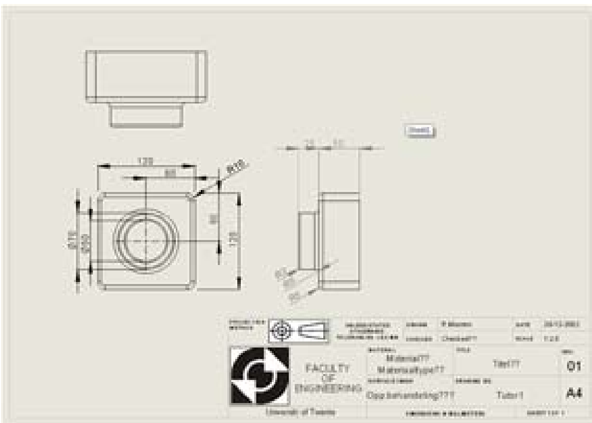
35. To prevent importing annotations that belong to hidden model items, deselect **Include dimensions from hidden features** in the **Options** box and click **OK**. Annotations on features that are completely hidden by other geometry will not be imported. This makes the import operation slower, but the resulting views do not contain annotations that you may not want.

    The dimensions are added to the **Front** view. Only the dimensions that are visible from the selected view are added. If these steps were repeated for the other views, only the dimensions which were not already shown would be added. Alternatively, you could use the the **Import Items into All Views** option in the **Model Items** PropertyManager to add the dimensions to all the views.

36. Drag the dimensions to position them orderly.

    Once dimensions have been added to a view, there are several options as to how they can be manipulated:
    - Drag dimensions by their text to new locations. Use the inference lines to align and position them.
    - Select the dimension by its text and press the **Delete** key. The dimension will be removed from the drawing sheet, but not from the model's database. Some dimensions are not required in the view. Therefore they can be removed. Dimensions that should appear in other views can also be deleted and added to the proper views.
    - Using **Shift**-drag to move, or **Ctrl**-drag to copy dimensions into other views.

    The second way of dimensioning a model is to add the dimensions manually. This will be explained now.



37. Select the **Right** view. Zoom to the view.

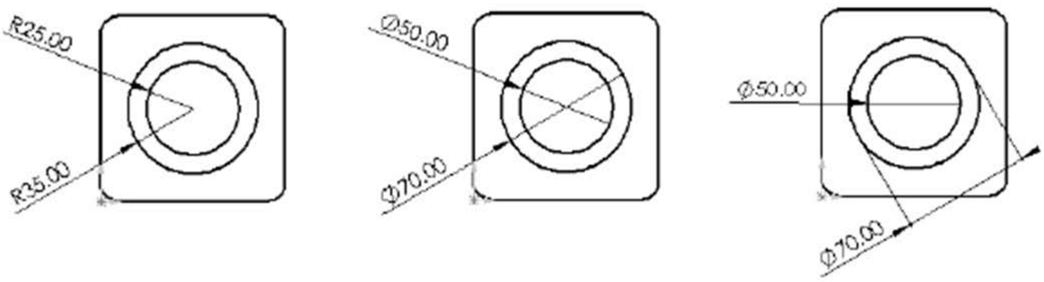
38. Click the **SmartDimension** tool in the **Annotation** tab.

39. Add the missing dimensions. The end result should look like the drawing shown to the right.

40. Save the file.

Dimensioning Tips

- To remove an unwanted dimension, select it and press the **Delete** key.
- To *hide* a dimension, click **View, Hide/Show Annotations** on the menu bar, then click the dimension(s) you want to hide.
- To *move* a dimension to another view, click the dimension, hold the **Shift** key, and drag the dimension to the desired location within the destination view boundaries (Do not drag by the handles when doing this).
- To *copy* a dimension to another view, click the dimension, hold the **Ctrl** key, and drag the dimension to the desired location within the destination view boundaries (Do not drag by the handles when doing this).
- To *center* the dimension text between the witness lines, right-click the dimension, and select **Display Options, Center Dimension**.

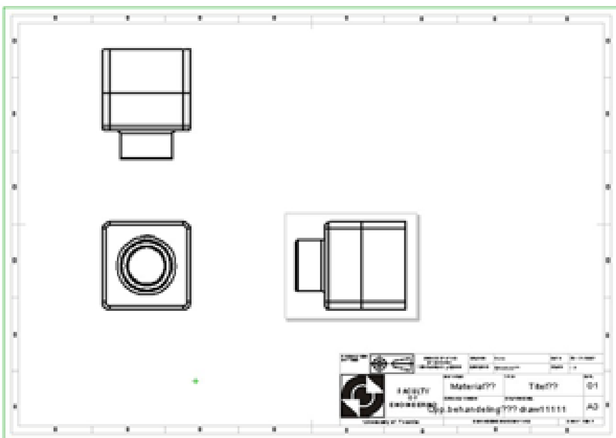


- For dimensions on circular features, you have these options:
  - To change a *radius* dimension to a *diameter* dimension, right-click the dimension, and select **Display as Diameter**.
  - To display a **diameter** dimension as a **linear** dimension, right-click the dimension, and select **Display as Linear**.
  - If the linear dimension is not placed at the angle you want, select the dimension, and drag the green handle at the attachment point. The dimension pivots around the circle, snapping in 15° increments.
- To modify the appearance of leaders, text, arrows, etc., right-click the dimension, and select **Properties**. Edit the available options, and click **OK**.
- To add reference dimensions in the drawing:
  - Click **Tools, Dimensions**, then choose a dimension type, or
  - Click **SmartDimension** and choose a dimension type from the right-mouse menu.
- To add annotations in the drawing:
  - Click **Insert, Annotations**, then choose the type of annotation to add, or
  - Choose a tool from the Annotations toolbar.
- To change the placement of dimension arrows in relation to the extension lines, click on the green point on one of the arrows, or click **Outside** , **Inside** or **Smart** in the **Dimension** PropertyManager.

For more information about adding and aligning dimensions and annotations in drawings, refer to the **Detailing** chapter of the **SolidWorks Online User's Guide**.

Adding Another Drawing Sheet

Now you create an additional drawing sheet for the assembly, including the standard three views, and an isometric view.



41. Right-click the sheet tab at the bottom of the window, and select **Add**. A second option is to right-click **Sheet1** in the FeatureManager design tree and select **Add Sheet...**

42. In the **Sheet Setup** you can select the ut template again.

43. Click **Standard 3 View** and click **Browse** in the PropertyManager to insert '**Tutor.sldasm'** in the drawing.

44. Reposition the views on the sheet if needed.

45. If the drawing sheet is too big or too small, you can choose a different scale or a different size. Right-click in a blank area of the drawing window (not inside the boundaries of a view) and select **Properties**. Select a different **Paper Size** and/or **Template** and/or **Scale**. Click **OK** or press **Enter** to view the changes.

Inserting a Named View

Named Views are views which take the orientation and name from the **View Orientation** dialog in parts and assemblies. All standard views, user named views and the current view are all eligible for use as a named view on a drawing sheet.

If the named view in the model is a perspective view, that information is carried to the drawing view. There are two ways to create a named view:

- use the tool on the **View Layout** tab or
- use the pull down option **Insert, Drawing View, Model....**

You can add named views to drawings, showing the model in different orientations. You can use:

- standard view (**Front, Top, Isometric**, etc.),
- a named view orientation you defined in the part or assembly,
- the current view in the part or assembly document.

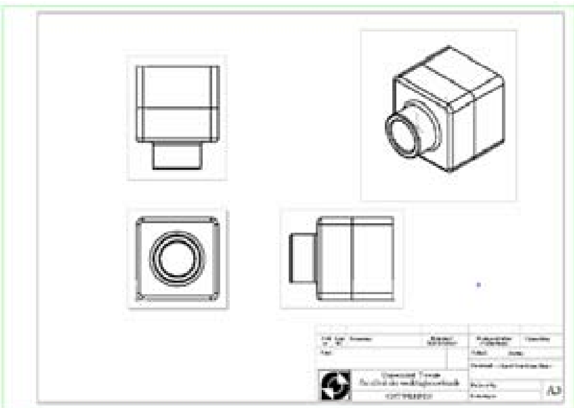
Zoom levels are ignored, however, and the entire model is always displayed in the selected orientation.

In this section you add an isometric view of the assembly.

46. Click or **Insert, Drawing View, Model....** The cursor shape indicates that you may select a model to display in the drawing. The **Model View** PropertyManager appears.

47. Select the **Tutor.sldasm** model in the **Open Documents** list

48. Click **Next** . Select **Isometric** from the **Orientation** list. The cursor shape indicates that you may select a location in the drawing to place the model view.



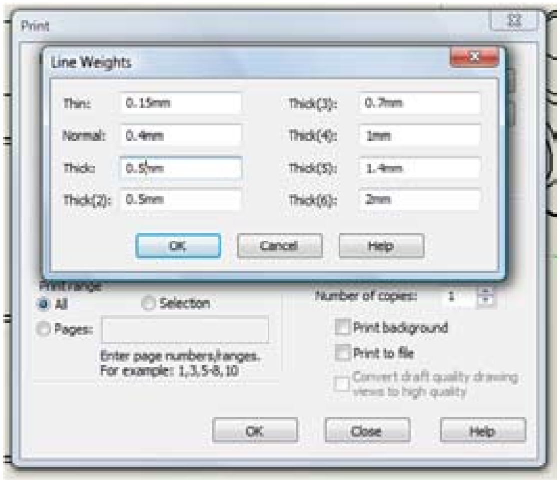
49. Click where you want to place the view in the drawing window and click **OK** in the **Model View** dialog.

Note: *isometric views cannot be used for dimensioning and indicating annotations. It is only a view for illustration.*

Notice that as you move the cursor into the region inside a view boundary, the boundary highlights and the cursor shape changes.

50. Save the drawing.

Printing the Drawing



51. Click **File, Print**. The **Print** dialog box appears.

52. Specify the print parameters, such as **Print range, Page Setup**, etc.

53. Under **System Options** select **Line Weights**.

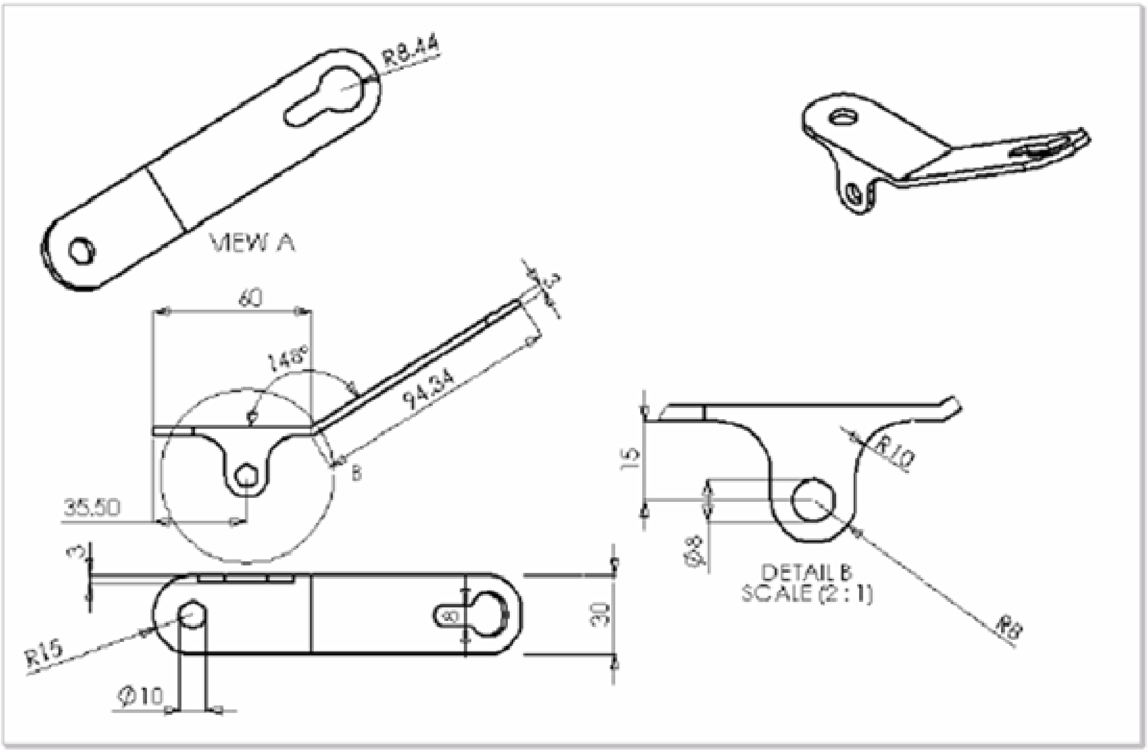
54. Set the thickness of **Thin** lines to **0.15 mm**, of the **Normal** lines to **0.4 mm**, and of the **Thick** lines to **0.5 mm**.

55. Click **OK** twice and collect the printed paper from the printer.

56. Close the drawing.

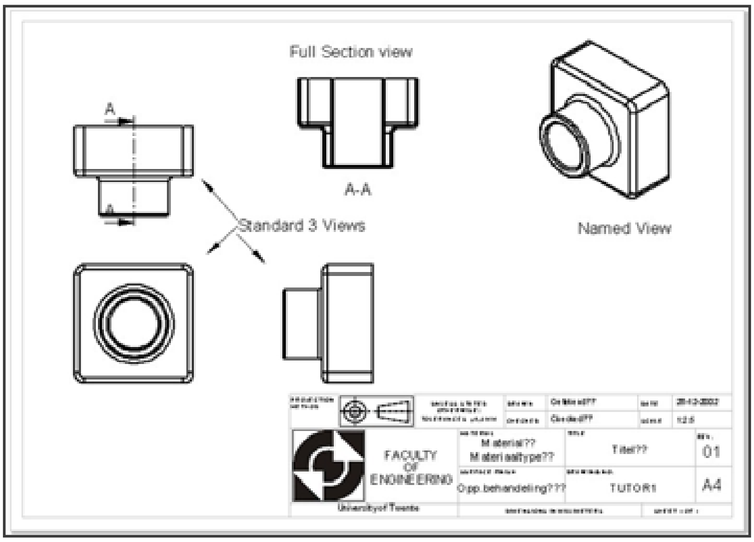
Advanced Drawings

In this part of chapter 4, you will learn how to make more advanced drawings and how to add annotations to them.



Views and Driving Dimensions

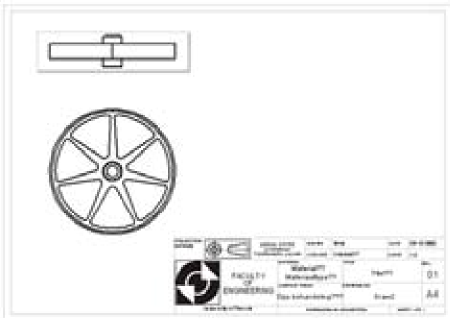
This section concentrates on the creation of drawing views and the addition of section views to the drawing.



1. Open the part **‘wiel_assy.sldasm’**, from the folder ‘**Chapter 4/Kruiwagen**’. To this part annotations will be added in the drawing.

2. Use **File, Make drawing from Part/Assembly** and use the view palette to insert the **Front** view of the assembly you have just opened into a drawing with a **drawing-ut-a3-hor** template. Make sure that the complete assembly is depicted in the view.

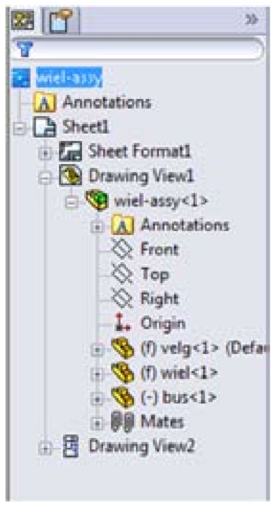
3. Add a **Top** view by moving the cursor above the Front view and clicking in the drawing window.



4. Save the drawing as */work directory/* **wiel_assy.SLDDRW**.

FeatureManager in Drawings

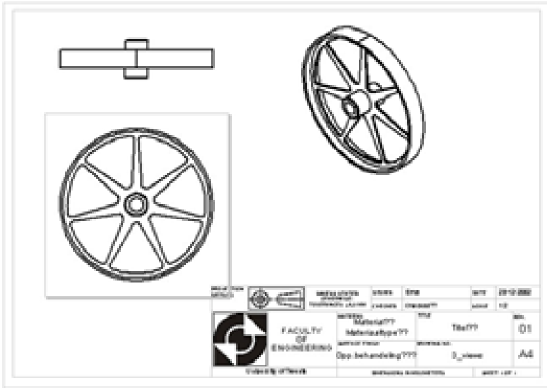
In the drawing file, the FeatureManager holds **Drawing Sheet**, **Template** and **View information**. Conceptually, these are similar to features in a part.



Each drawing file has one or more drawing sheets. In this example, it has the dummy name **Sheet1**. The name can be changed at any time using the Properties of the sheet. The p**roperties** also include view scaling and section numbering.

5. Expand the drawing sheet **Sheet1** and see all the views. Expand **Drawing View1**.

Each **Drawing View** contains a reference to a part or an assembly. In this example, the reference for both of the views is the assembly **wiel_assy.** The reference name can be expanded to display the entire part. Selecting a drawing view in the FeatureManager window is equivalent to selecting it in the graphics window.



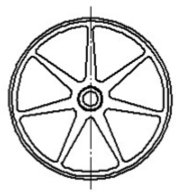
6. Click the **Model View** tool. Select **wiel_assy** in the **Open Documents** box and click **Next**.

7. Select ***Isometric** in the **Orientation** list.

8. Place the isometric view in the drawing as shown below.

9. Save the drawing.

Full Section View

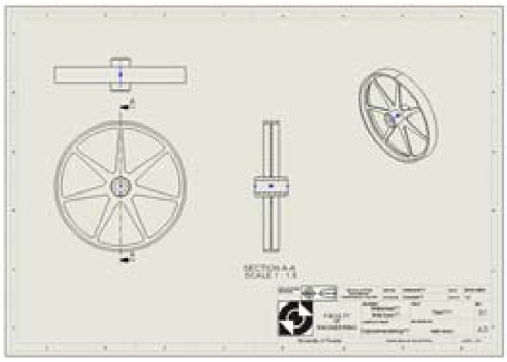


Several types of section views can be created. The Full Section View uses one or more lines and arcs to form the cutting surface. In this example, a single line section will be created.

10. In the **Front** view, sketch a vertical section line through center of the circle. Use the cursor feedback.

Section View creates a full or partial section view based on a cutting line and a direction. A single sketch line is used for the section line. You can make a section view:

• from the pulldown menu choose: **Insert, Drawing View, Section** or
• from the View Layout tab pick the tool.



You are now going to create a section view of the wheel.

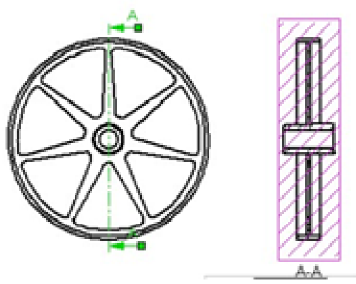
11. With the section line highlighted, click the **Section View** tool.

12. In the **Section View** dialog, select **Auto hatching** and click **OK**.

13. When the view appears, move to the right and locate the view as shown to the right.

The drawing view **Section View A-A** is aligned to the **Front** view and comes with a label beneath it, if necessary name the label Section A-A by double-clicking the label. The section includes **Axes** created where the section cuts a circular face. These axes can be hidden, resized or deleted.

Once the section appears, the arrows will face in a default direction. They can be reversed by simply double-clicking on the section line. In many cases, this causes the view to appear crosshatched in red. This indicates that the view has been changed and needs to be updated.

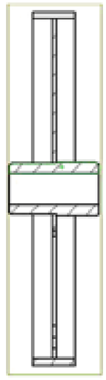


14. Double-click the section line.

The arrows will reverse their direction and the section view appears crosshatched as shown to the right.

15. Update the view using **Rebuild** .

Now that the Rebuild crosshatch is gone, we will adjust the normal crosshatch of the drawing.



A-A

*Note: most of the time **Auto Hatching** give sa good result right away.*

16. Zoom in on a piece of the section view with crosshatching.

17. Click a crosshatched area of the part **bus.** The **Area Hatch/Fill** PropertyManager appears.

18. To adjust one area, deselect **Material crosshatch**. The options are now no longer grayed out.

19. Change the **Scale** of hatching to **2.0**. Change the **Angle of hatching** to **90.00** ° to give it an obvious other angle that the other crosshatched areas. click **OK**.

*Tip: If the changes are not shown in the graphics area while changing the data, select **Apply changes immediately** in the **Opti-ons** box.*

20. Click **Rebuild**.

View changes, whether caused by changes to the view or to the part/assembly it references, can be handled in several ways by **Automatic View Regeneration** which is turned on by default. Depending on the settings used, the regeneration of a modified view may be delayed or be automatic when the change occurs. The options are:

- **Global setting**
  Right-click the drawing icon at the top of the FeatureManager design tree and select **Automatic View Update**.
- **All views**
  The entire drawing can be Rebuilt, by clicking on **Edit, Rebuild** or  , causing all out-of-date views to be updated at once.

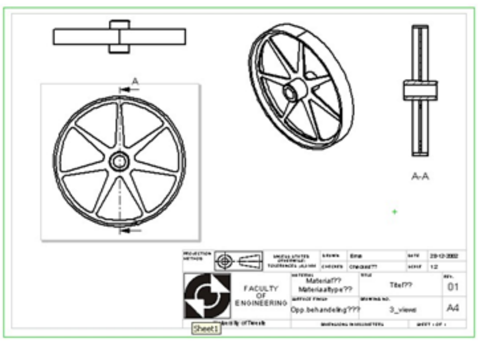
Breaking View Alignment

The alignment of (section) views can be broken or restored at any time. The new section view is currently aligned to the front view. That alignment should be broken and the view should be moved.

21. Select the view and choose the option **Alignment, Break Alignment** from the right-mouse menu.
The view will be released from the default alignment setting.

22. Drag the section view to the right of the isometric view as shown below. Note that the **Section A-A** label moves with the view.

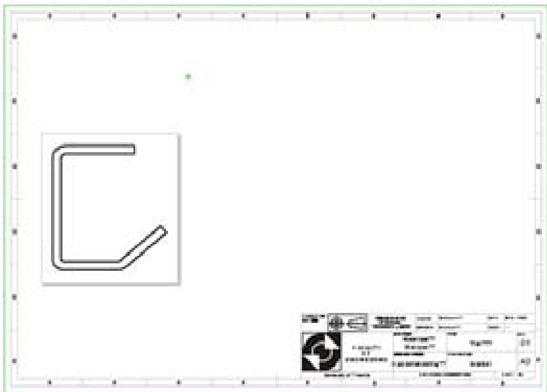
23. Save the drawing.



Auxiliary, Projected and Detail Views

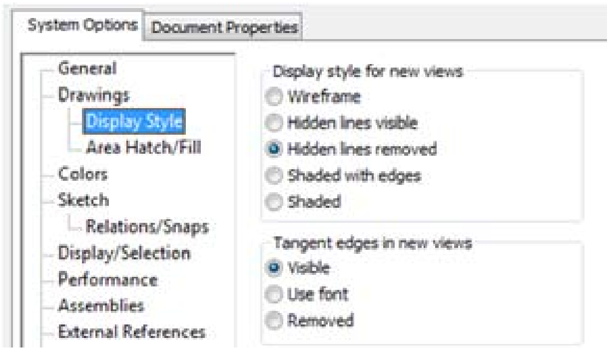
The **Auxiliary** and **Projected Views** commands can be used to create views from existing views. Detail views provide the ability to focus on an area delineated by a sketched shape, while expanding the scale.

24. Open the part **'bracket.sldprt'** (also available in the 'Chapter 4' directory) and create a drawing of the **Left** view using the **Make drawing from Party** button with the template **drawing_ut_a3_h**. Set the scale to **1:1**. Use the view palette to drag the **Left** view to the sheet.



25. Change the settings for the display of new views in the drawing. Under **Tools, Options, Drawings, Display Style** set the **Display style for new views** to **Hidden lines removed**. Individual views can be changed to any of these three options after they are created.

__Tip:__ it is much easier to select the different views one at a time and click one of the display type buttons in the __View__ toolbar: __Wireframe__, __Hidden lines visible__ or __Hidden lines removed__.



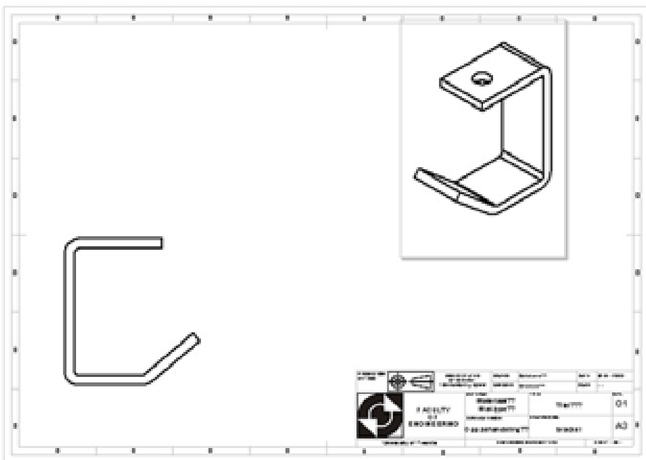
Copy and Paste a View

Views can be copied and pasted to the current drawing sheet or another sheet. Use standard tools to copy and paste the selected view. Use the FeatureManager design tree to paste to a drawing other than the current one.

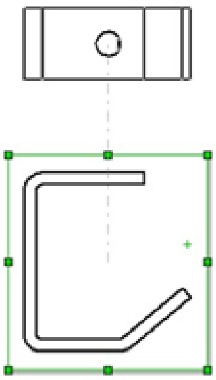
26. Select the **Left** view and use **Ctrl+C, Edit, Copy** or the **Copy** tool to copy the view to the clipboard.

27. Click in the drawing. Use **Ctrl+V, Edit, Paste** or the **Paste** tool to place the copy onto the upper right corner of the drawing. The new view is an exact duplicate of the original, but it can be changed.

28. Select the new view and double-click ***Isometric** in the PropertyManager to change the orientation of the view. Click **OK**.



Projected View



A Projected View is a quick way to create a view from an existing one. The view is projected and aligned in one of four directions surrounding the existing view. A projected view is created in one of the following ways:

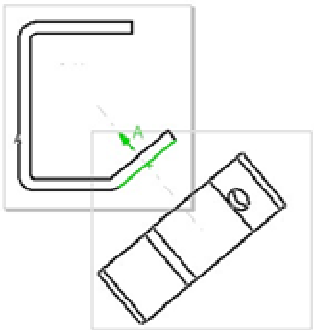
- from the pull down menu choose: **Insert, Drawing View, Projection** or,
- from the View Layout tab pick the tool.

29. Select the front view and insert a **Projected View**. Moving the cursor around the four sides of the source view will change the preview picture. Place the new view above the source with a click.

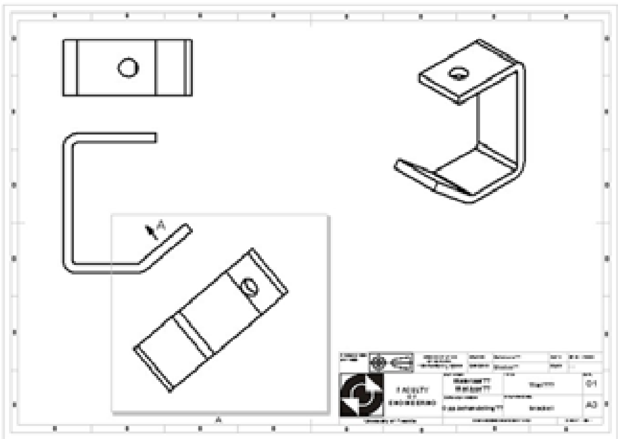
Auxiliary View

Auxiliary View creates a new view as a projection from a selected edge. The new view is aligned to the source. A View Arrow and label are also added to the display. The view arrow shows the direction of the view with a label. In this example, the auxiliary view will be used to create a view of the angled flange. You can find it:

- from the pull down menu choose: Insert, Drawing View, Auxiliary or,
- from the View Layout tab pick the tool.



30. Select the angled edge of the model in the front view and click the **Auxiliary View** tool. The preview is aligned to the source, normal to the selected edge.



31. Locate the view at the bottom of the drawing, as shown below.

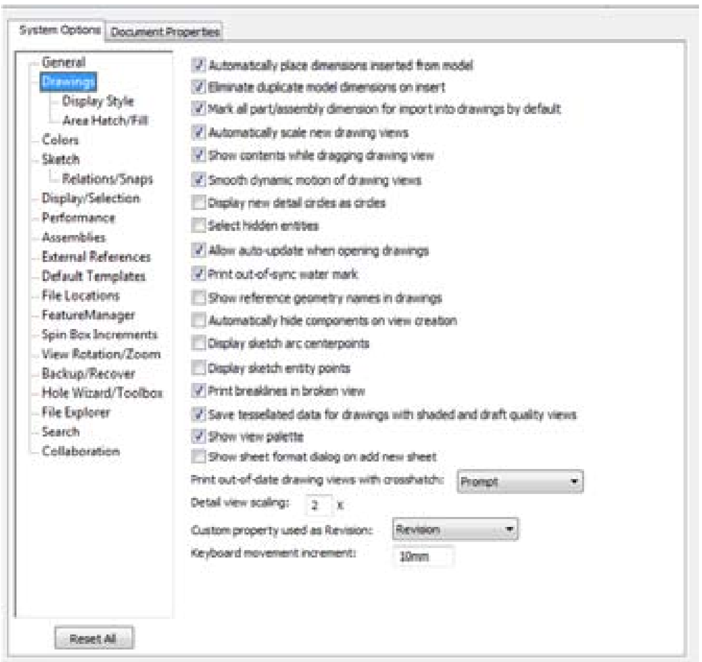
    The view is located and a view arrow is added to the drawing. By double-clicking the view arrow, the orientation of the auxiliary view is changed. The note displays the same letter as the view arrow, in this case, **A**.

Detail Views

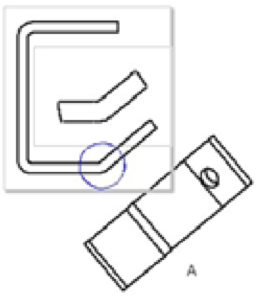
Detail Views can be created using a closed sketched shape in an activated source view. The detail can use a scale multiplier to scale it n times larger than its source. The content of the detail view is determined by what is enclosed within the sketch. There are two ways to achieve it:

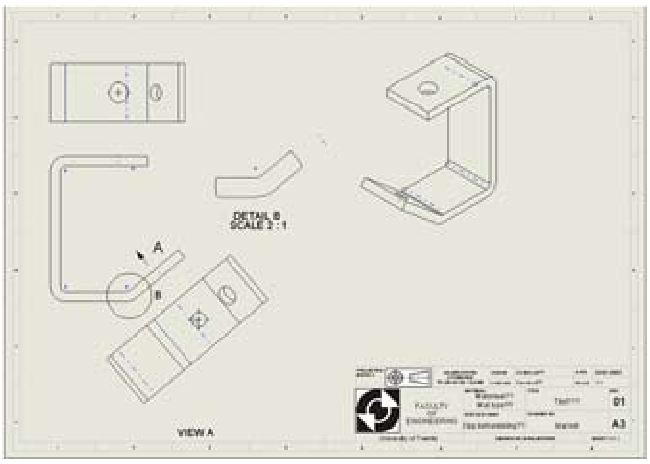
- from the pull down menu choose: **Insert, Drawing View, Detail** or,
- from the View Layout tab pick the (A) tool.

32. Using the **Tools, Options, Drawings** dialog, set the scale multiplier for detail views. Set the **Detail view scaling** to **2**.



33. Select **Detail View** and draw a circle around the non-perpendicular bend in the ***Left** view as shown below. The view is created at the moment the circle is completed.





34. Move the cursor and place the detail view as shown below by clicking the left mouse-button.

35. Click **OK** to set the tangent edges display to **Visible**.

36. Select the edge of the circle and reduce the diameter of the circle by dragging the edge inward.

37. Save the drawing in your */work directory/* as **bracket** and close it.

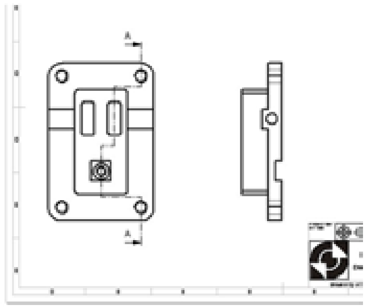
Offset Section Views

Section views are not restricted to single line sections. Multiple lines and arcs can be used to form the section line.

38. Insert a **Front** view of the model **'Offset section.sldprt'** from the **'Chapter 4'** directory in a drawing with a template **drawing-ut-a3-hor**.

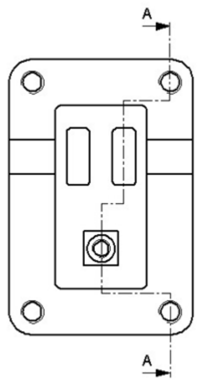
39. Use the **Projected View** tool to create a **Right** view, as shown to the right.

40. Right-click the drawing sheet name and select **Properties** from the menu.



41. Change the name of the drawing sheet to **Offset Section**.

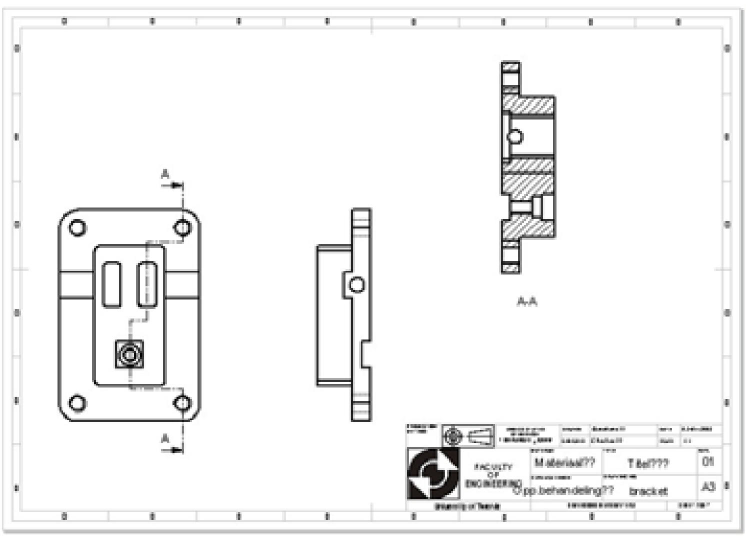
42. Draw a section line as shown below using references. The section line should pass through the centers of the three holes and through the center of the slot.



43. Click the **Section View** tool to create the offset section. Position it to the **left** of the **source** view. **Reverse** the direction by double-clicking the section line and rebuilding the view.

44. Right-click the view and select **Alignment, Break Alignment** from the drop-down list.

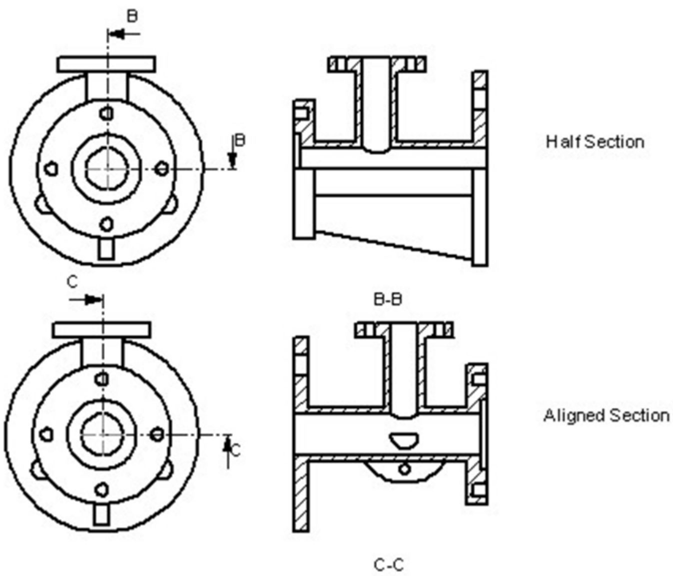
45. Position the section view in the right top corner of the drawing sheet as shown below.



46. Save the drawing in your */work directory/* as **Offset_section** and close it.

Aligned and Half Section Views

Some other versions of the section view can be created. The half section uses the standard Section tool, while the aligned section rotates circular sections into a common plane.

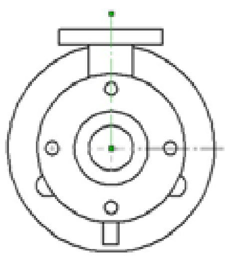


The half section view displays a section on one half and the unsectioned model on the other. It is created using the standard section view tool.

47. Open the part **‘spruitstuk.sldprt’**, available in the ‘**Chapter 4**’ directory.

48. Create a **Model View** of **spruitstuk.sldprt**, with a **Top** orientation in a drawing with template **temp_ut_a3_h**.

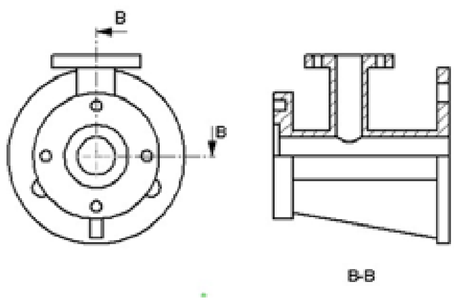
49. Change the name of the sheet to **Half and Aligned**.



50. Draw horizontal and vertical section lines through the center of the model as shown below. To make sure the lines come together in the center of the circle, select **Tools, Sketch Settings, Automatic Solve**.

51. Select the vertical centerline and click the **Section View** tool to create the section view. A message box appears.

52. Click **No** to create a section through the whole part. It is aligned to the source view. Position it to the right of the view.



The **Aligned Section** tool is used when the cutting plane is bent to include certain angled elements. The resulting section is then revolved into a single plane. In this example, the section will be set up to pass through one hole. There are two ways to find it:

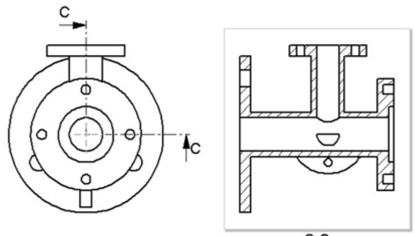
• from the pulldown menu choose: **Insert, Drawing View, Aligned Section**.
• from the View Layout tab, click the downward arrow under **Section View** and pick **Aligned Section**

53. Create another **Top** view of **spruitstuk.sldprt** by copying and pasting the first one.

54. In the second Top view the section lines of the first one are still visible. Select the section lines and delete the **Section View**.

55. Select the vertical line to define the alignment of the new view. Add an **Aligned Section View** by using one of the options mentioned above. The new view is aligned to the right of the source.

56. Save the drawing in your */work directory/* as **Spruitstuk**.

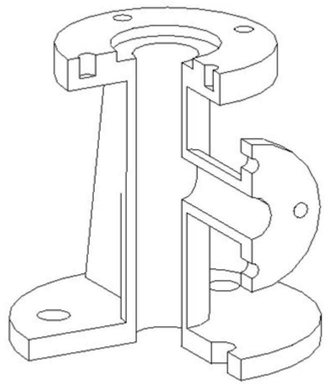


C-C

Cutaway Views

Cutaway views of parts and assemblies can be created in SolidWorks. This type of section is created in the part or assembly as a cut or assembly feature and then it is displayed in the drawing. The crosshatching is not automatic for these views. It is added in the drawing using **Area Hatch**.

SolidWorks has the ability to represent and store parts and assemblies in more than one version. These versions are called **Configurations**. In this case we have two configurations of the housing - one with the cutaway and one without it. Which configuration appears in a particular drawing view is controlled through the view's properties.

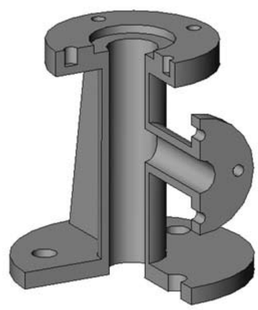

Cutaway View

57. Add a drawing sheet named **Cutaway** by right-clicking the tab **Sheet1** at the bottom of the drawing window.

58. Reopen **spruitstuk.sldprt**.

A cut feature was created to generate the cutaway shape required. It is named **section cut** and appears in the FeatureManager design tree. Notice that it appears gray. This is because it is suppressed.

59. Click the **configurationsmanager**  at the top of the FeatureManager design tree.



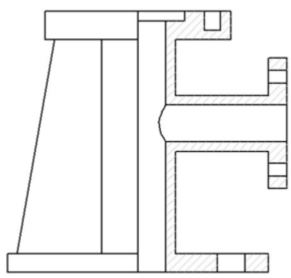
60. Double-click on the configuration named **Cutaway**. This configuration shows the cut feature. The Default configuration does not.

61. From the **Window** menu, choose **Spruitstuk - Cutaway**.

Tip: *use* ***Ctrl+Tab*** *to quickly cycle through the open documents.*

62. Insert a **Standard 3 View** of the Cutaway configuration of **spruitstuk**.

To change which configuration is shown in a drawing view, edit the view's **Properties** in the right-mouse menu. Change the **Use named configuration** listing. Click **OK** and **Rebuild** to change the configuration of the view. Since we selected the Cutaway configuration in the part, the right configuration is automatically selected.



63. Add hatching to the cut faces of the model. **Ctrl**-select the faces which require crosshatching,

64. Right-click on one of the areas, select **Annotations**, **Area Hatch/Fill**. Alternatively, pick **Area Hatch/Fill** from the **Insert, Annotations** menu. The **Area Hatch Fill** PropertyManager appears.

65. Check if the right areas are selected and click **OK.**

    Note: notice that no preview is shown.

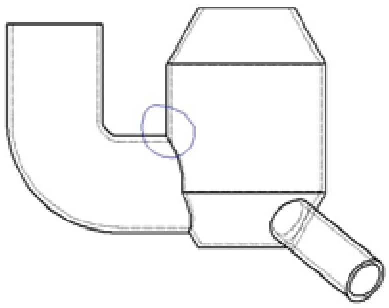
66. Save the drawing and close it.

Broken Section Views

Broken sections, like cutaway views, involve the use of cut features and configurations to control their display.

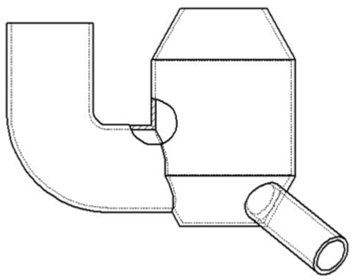
67. Open the part **'Verdeelstuk.sldprt'**, created in chapter 2 and create a **Bottom view** of the part in a drawing with a template temp_ut_a3_hor.

68. Select the **Broken-out Section** tool and draw a spline as indicated to the right.



69. Set the **depth** to **90 mm** and click **OK**. The view has an automatic crosshatching.

70. Save the drawing in your */work directory/* as **Sweep**.



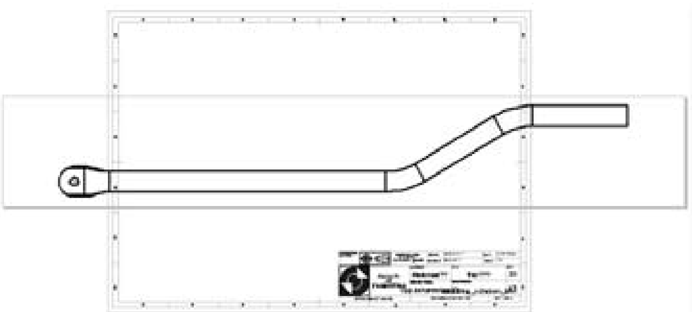
Broken Views

Sets of horizontal or vertical break lines can be added to a view. These break lines foreshorten the view of the model leaving only a small gap between them.

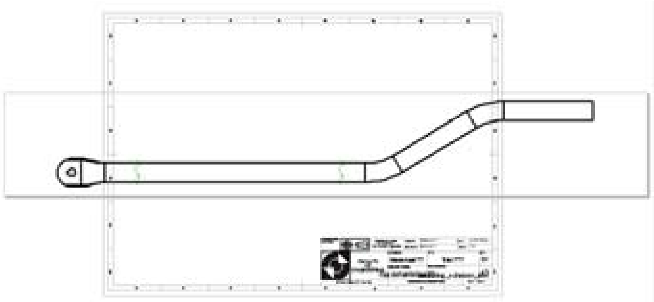
71. Open the part **buis.sldprt** from **'Chapter 4/Kruiwagen'** and create a **Front** view of the part in a new drawing with the template **temp_ut_a3_hor**.

72. Set the scale to **1:1**.

    In this example, the standard view is too long for the drawing sheet.



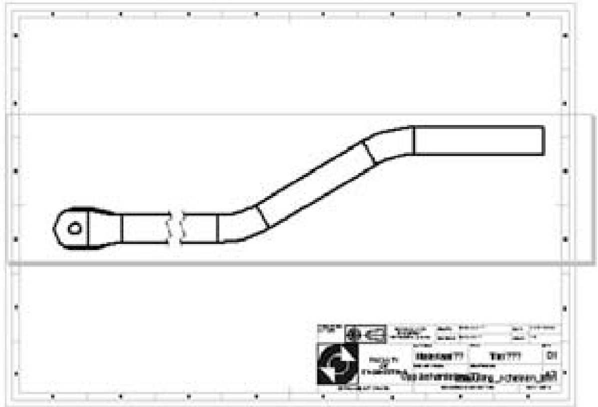
73. Break line sets can be inserted in horizontal or vertical orientations. Select the view and choose **Break** from the **Drawings** menu. The break lines are added to the view.



74. Drag and drop the cuts to position them within the view.

75. The break lines appear as **Zig Zag Cuts** by default, but they can be changed. Select one of the lines and use the pull-down menu from the feature manager to choose **Straight Cut, Curve Cut** or **Small Zig Zag Cut**.

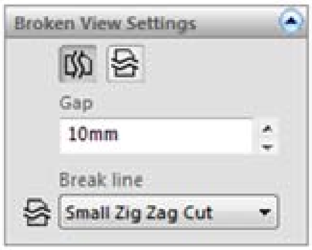
Important: *if multiple sets of break lines are used, they must all be positioned before the view is broken.*



76. Select the view and pick **Break View** from the right-mouse menu. The view will be foreshortened between the break lines.

77. Save the drawing in your */work directory/* as **buis**.

The distance between the break lines is controlled in the **Broken View Settings** of the feature manager, by selecting the break and changing the value of the gap.



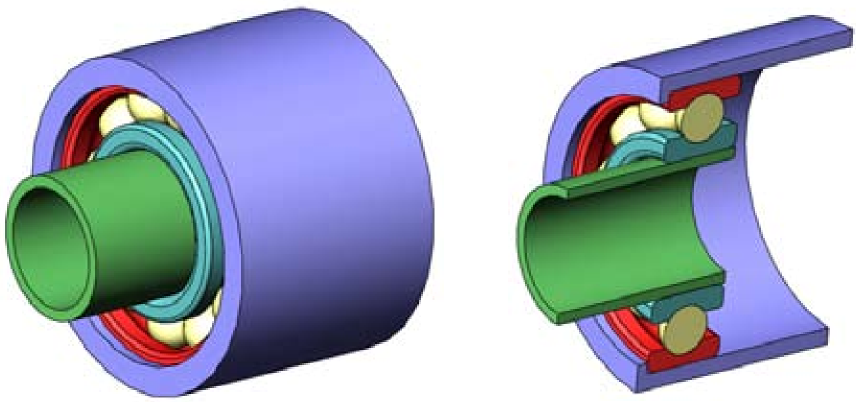
Even after the view is broken, the break lines can be moved. When a break line is being dragged, the entire model appears temporarily. When the break line is dropped, the view returns to its broken state and updated position.

78. Close the drawing.

Assembly Section Views

Making a section view of an assembly requires the same steps as for a part. The only difference is that you can control which components of the assembly are affected by the section and which are not.

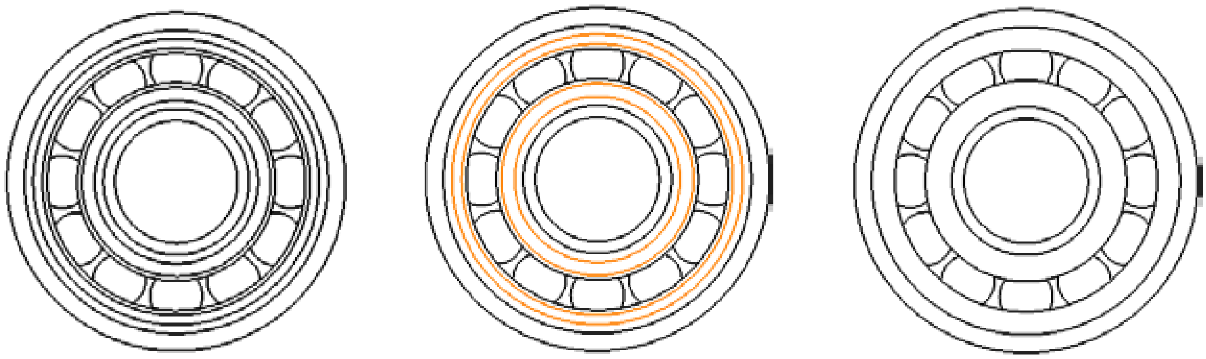
79. Open the assembly '**groefkogellager**' from the '**Chapter 4/kogellager**' directory.



80. Create a drawing of the **groefkogellager** with a **temp_ut_a4_hor** template. Insert a **Right** view and set the Sheet **Scale** to **1:1**.

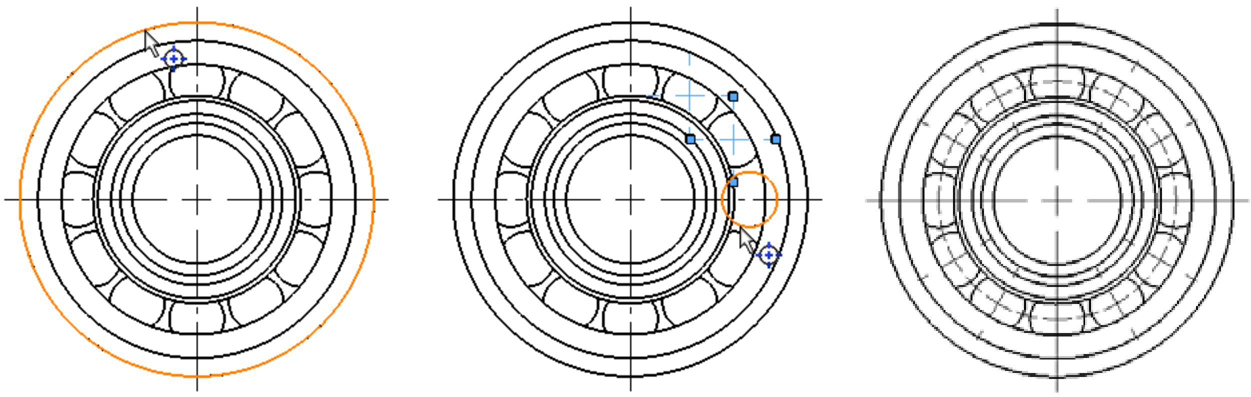
81. The parts in the ball-bearing have a lot of fillets. to clarify the drawing, we will hide some of these edges. Right-click the drawing and select **Hide/Show Edges** . The **Hide/Show Edgesb** PropertyManager appears.

82. From the outside to the inside, hide ring number 3, 4, 7 and 8, as shown below. Click **OK**.



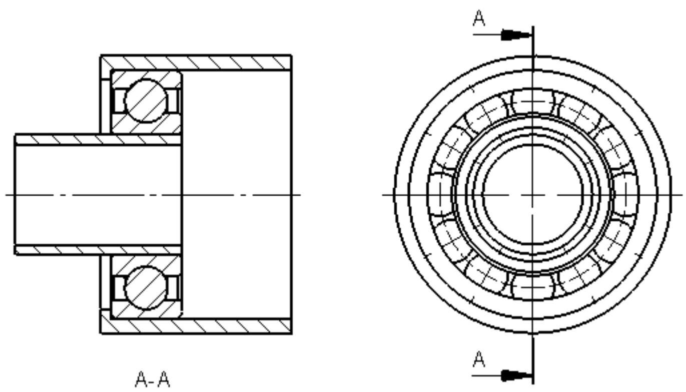
The next step is to create centermarks for the balls and the rings of the ball-bearing.

83. Turn on the **Center Mark** tool. Create a Center Mark for the outer circle. Under **Options**, set the type to **Circular Center Mark**. Deselect the **Circular lines** option and turn on **Radial lines**. Select the balls by clicking on their edges one by one. SolidWorks will detect the pattern and will create a round circle. Click **OK**.



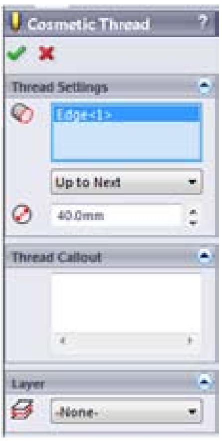
84. Sketch a vertical line in the middle. Highlight the geometry and click the **Section View** tool . The **Section Scope** dialog appears. If necessary, flip the Section View by double-clicking the section line and rebuild the view to reverse it. Add a centerline through the middle and Center Marks to the balls.

SolidWorks automatically alternates the angle of similar crosshatching on adjacent components. The crosshatching can also be changed by editing its properties.



85. Save the drawing as **Groefkogellager** drawing in your */work directory/*.

Annotations



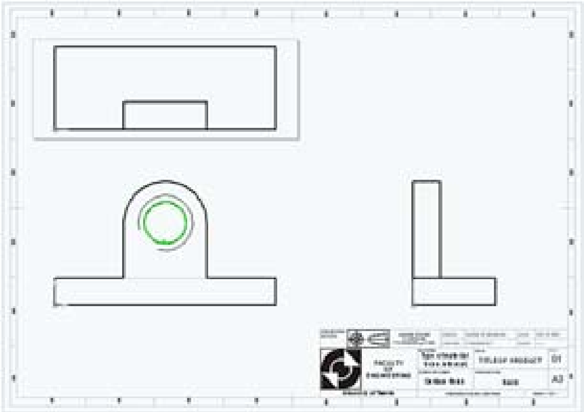
86. Open the drawing **'Basic.SLDDRW'**. It can be found in **'Chapter 4/Basic'**.

87. Select **UpperHole** of **Basic** in **DrawingView1**.

88. Click **Insert, Annotations, Cosmetic Thread...** The Cosmetic Thread dialog box appears.

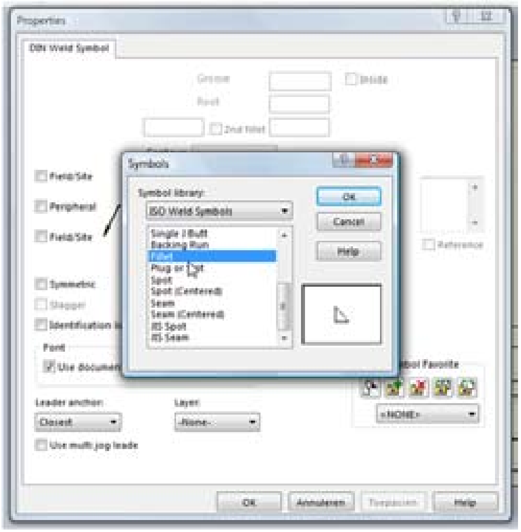
89. Set **Cosmetic Thread** to **Blind**, **14.0 mm** and the **Major diameter** to **40.0 mm**.

90. Click **OK**.



91. Select the right bottom corner of **Vert_Boss** of Basic in the **Drawing View1** (to see where the weld symbol should be positioned, you can take a look below).

92. Click **Insert, Annotations, Weld Symbol...** The **Properties** dialog box appears.

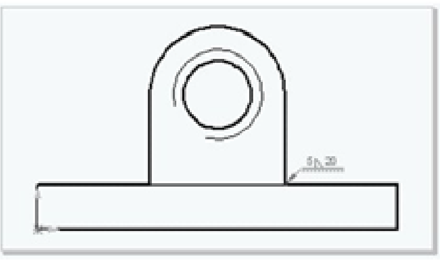


93. Click the button **Weld symbol** above the weld arrow and select **Fillet** from the Symbols list.

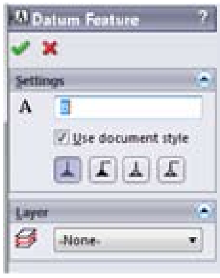
94. Set the box to the **left** of the button **Weld symbol** to **5**.

95. Set the box to the **right** of the button to **20**, which represents the length of the weld.

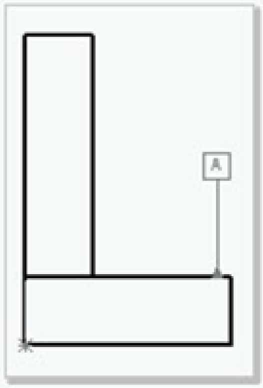
96. Click **OK**. The Weld Annotation is placed. Move it to a convenient location as shown below.



97. Select an edge of **BasePlate** of **Basic** in **DrawingView3**.

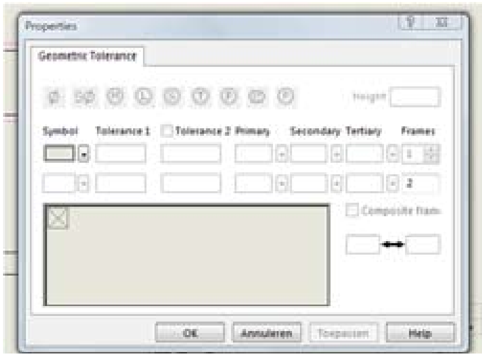


98. Click **Insert, Annotation, Datum Feature Symbol** or click on it on the Annotation tab. The **Datum Feature** PropertyManager appears.

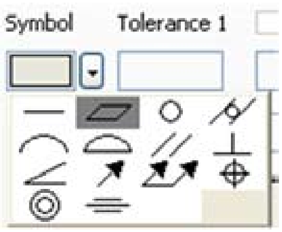


99. Click in the drawing window where you want to place the symbol and click **OK**.

100. Select the edge of **BasePlate** of **Basic** again in **Drawing-View3**.

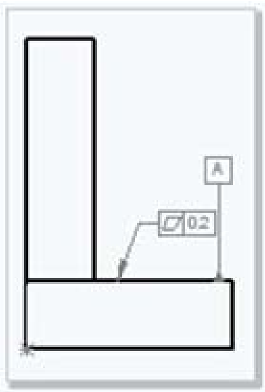


101. Click **Insert, Annotations, Geometric Tolerance** or click on it on the Annotation tab. The **Properties** dialog box appears.



102. Click **Symbol**.

103. Select **Flatness**.



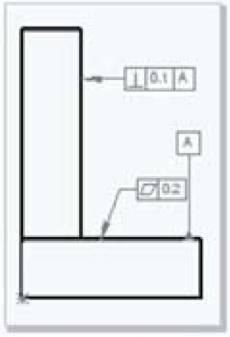
104. Set **Tolerance 1** to **0.2** and click **OK**. Reposition the Tolerance.

105. Select the edge **Vert_Boss** of **Basic**.

106. Click again **Insert, Annotations, Geometric Tolerance...** The **Properties** dialog box appears.

107. Click **Symbol** and elect **Perpendicularity**.

108. Set **Tolerance1** to **0.1**, **Primary** to **A** and click **OK**.

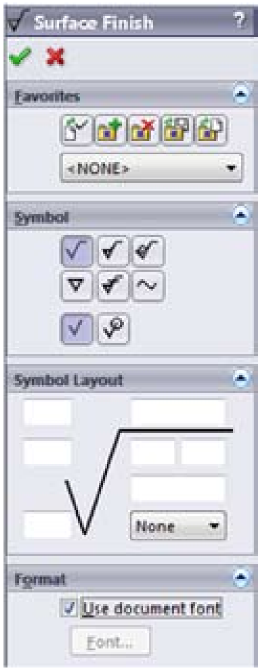


109. Click **Insert, Annotation, Surface Finish Symbol** or . The **Properties** dialog box appears.

110. Set **Symbol** to **Basic**.

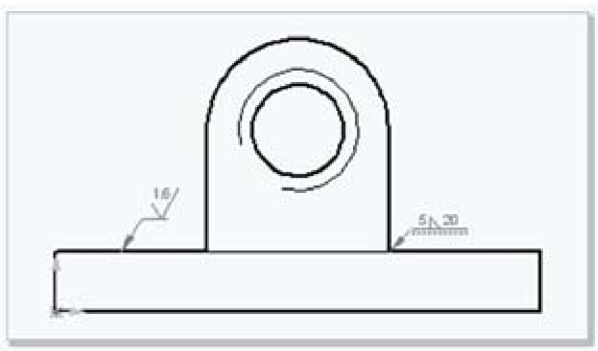
Note, You can also choose a different symbol. If you want the part to be made by a machine, you choose Machining. If you don't want the part to be made by a machine you choose non-machining. If you don't mind how the part will be made, or if you want both you choose Basic.

111. Set in the **Symbol Layout** box, the top left box to **1.6.** This indicates the roughness of the surface.

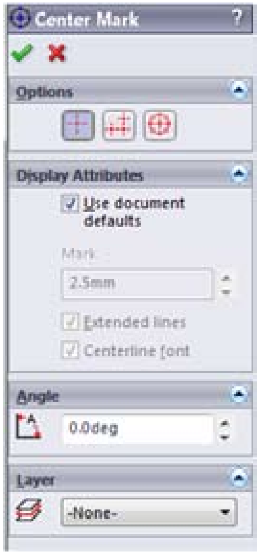


112. Select an edge of **BasePlate** of **Basic** in **DrawingView1**.

113. Click **OK**.



114. Select **Upperhole** of **Basic** in **DrawingView1**.

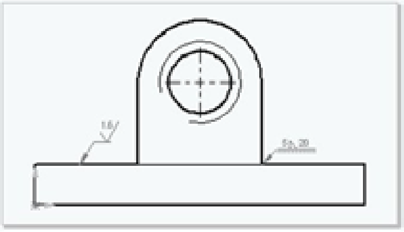


115. Click **Insert, Annotations, Center Mark...** or click . The **Center Mark** Property Manager appears.

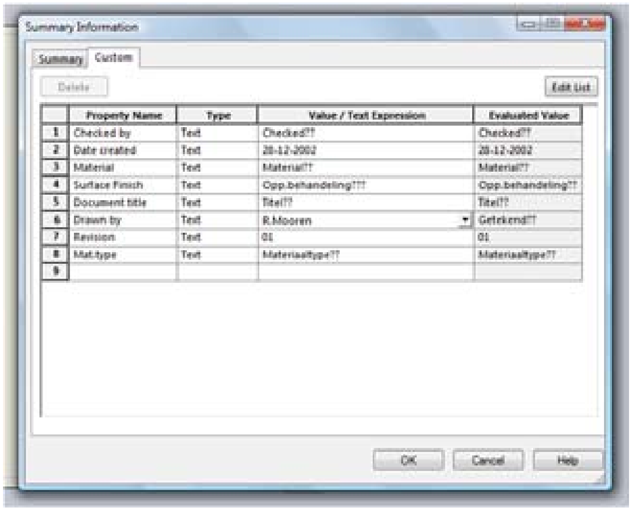
Note: *All holes should always be marked with a center mark!*

116. Select **Use document's defaults** and set **Angle** to **0.0 deg**.

117. Click **OK**.



118. Click **File, Properties...** The **Summary Information** dialog box appears.



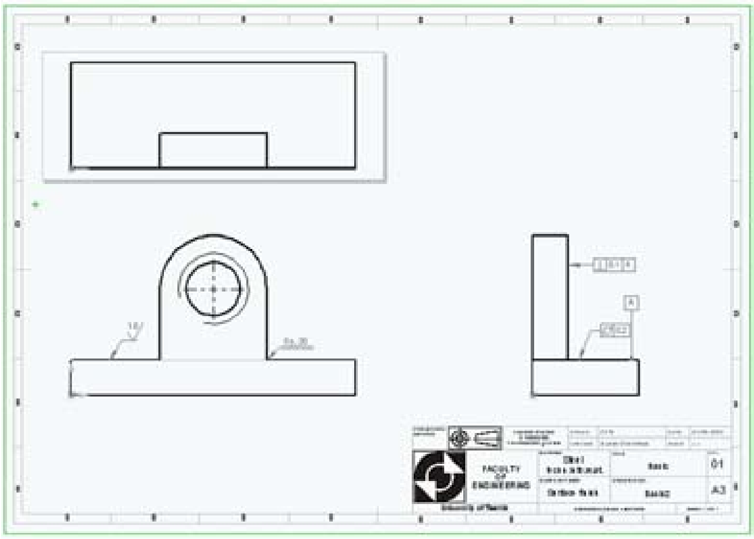
	Property Name	Type	Value / Text Expression	Evaluated Value
1	Checked by	Text	Checked??	Checked??
2	Date created	Text	20-12-2002	20-12-2002
3	Material	Text	Material??	Material??
4	Surface Finish	Text	Opp.behandeling???	Opp.behandeling??
5	Document title	Text	Titel??	Titel??
6	Drawn by	Text	R.Mooren	Getekend??
7	Revision	Text	01	01
8	Mat.type	Text	Materiaaltype??	Materiaaltype??
9				

119. Select the **Custom** tab.

120. Select **Checked by** at **Properties**. Enter your name at **Value**.

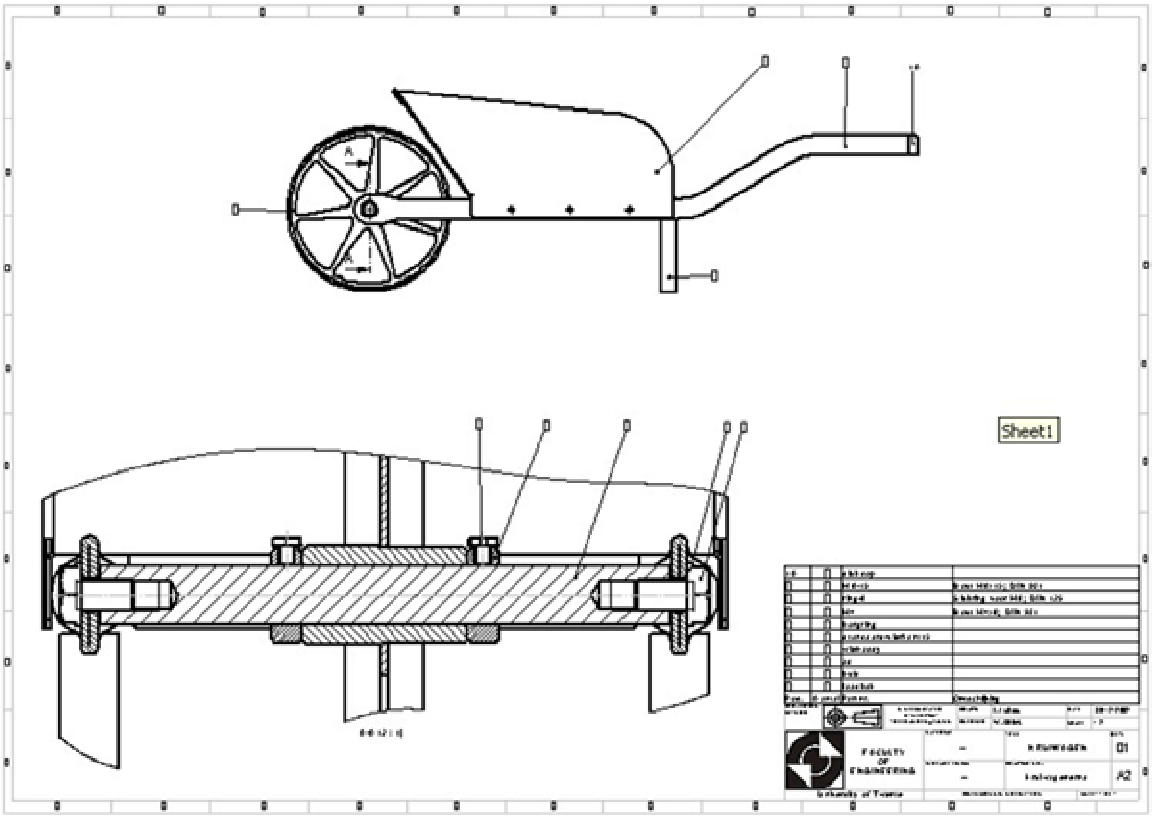
121. Fill in the other properties and click **OK**.

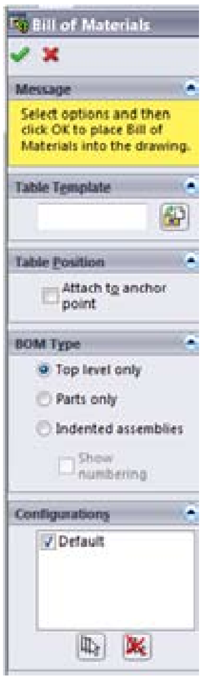
122. Save the drawing.



Bill of Materials

In this part of chapter 4, you add a Bill of Materials (BOM) and balloons to a drawing of a wheelbarrow assembly.





1. Open the drawing **kruiwagen.SLDDRW**, from the '**Chapter 4/Kruiwagen**' directory.

   Because a drawing can contain views of different parts and assemblies, you must pre-select the view for which you want to create a bill of materials.

2. Select **Drawing View3** and select **Bill of Materials** from the **Insert, Tables** menu. The **Bill Of Materials** PropertyManager appears.

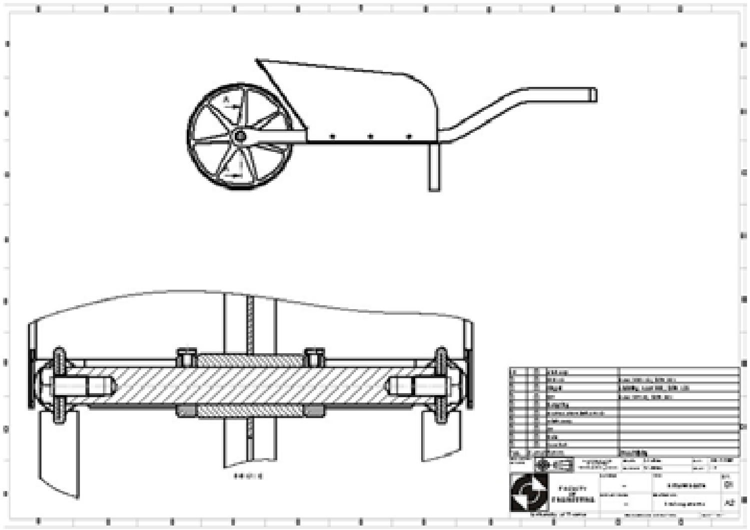
3. Click in the **Table template** box the **Open table template for Bill of Materials** button and select the bill of materials template **stuklijst-UT**. If it is not visible browse to the location of the file.

4. Set the following items:
   • **Top level only** – the parts and subassemblies are listed, but not the subassembly components

5. Click **OK** to close the **Bill of Materials Properties** PropertyManager and place the BOM above the title block.

   A bill of materials is displayed. It lists the parts and subassembly in the universal joint assembly. If you click in the BOM it activates automatically and the list can be edited.

6. Click in the cells of the last columns and add a description of the parts. Click outside the bill of materials to close it when you are done.



Balloons callouts label the parts in an assembly drawing and relate them to item numbers on the bill of materials.

7. Click **Balloon** on the Annotations tab, or click **Insert, Annotations, Balloon**.

8. Click a component in the drawing view. Click again to place the balloon.

   A balloon attaches to the component. The numbers correspond to the item numbers or quantities in the bill of materials. This can be chosen in the **Balloon** dialog under **Balloon text**.

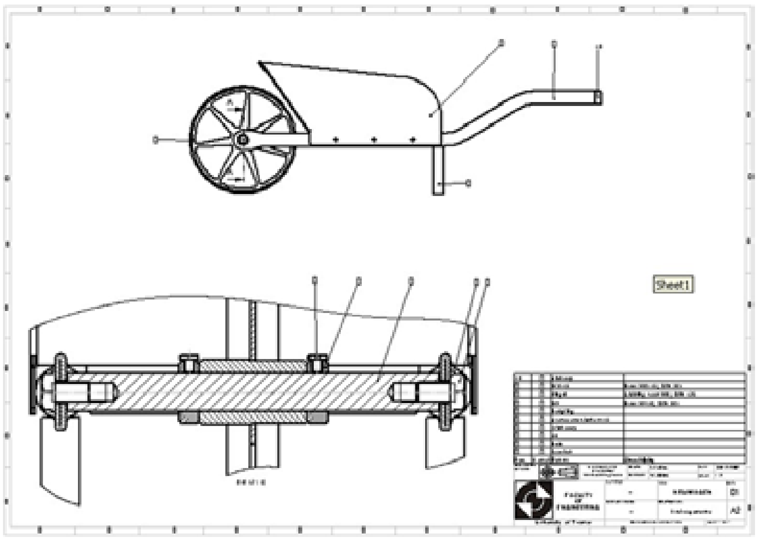
9. Click **Select** or press **Esc** to turn off the balloon tool.

10. To move the balloon or leader arrow, select and drag the balloon, or drag the leader by the handle.

11. Insert all the balloons necessary and place them in a clear order.

    You can save the bill of materials as an Excel file for use with other applications.

12. Select the bill of materials.



13. Click **File, Save As**.

    The **Save Bill of Materials Table** dialog box is displayed. Notice that the **Save as type** is set to **Excel Files (*.xls)** by default.

14. Type **Excenteraandrijving_BOM** in **File Name** box and click **Save**.

    Note: *The Excel file is not linked to the bill of materials in the drawing. If assembly components change, the bill of materials automatically updates, but the text file does not.*

15. Save the drawing also.

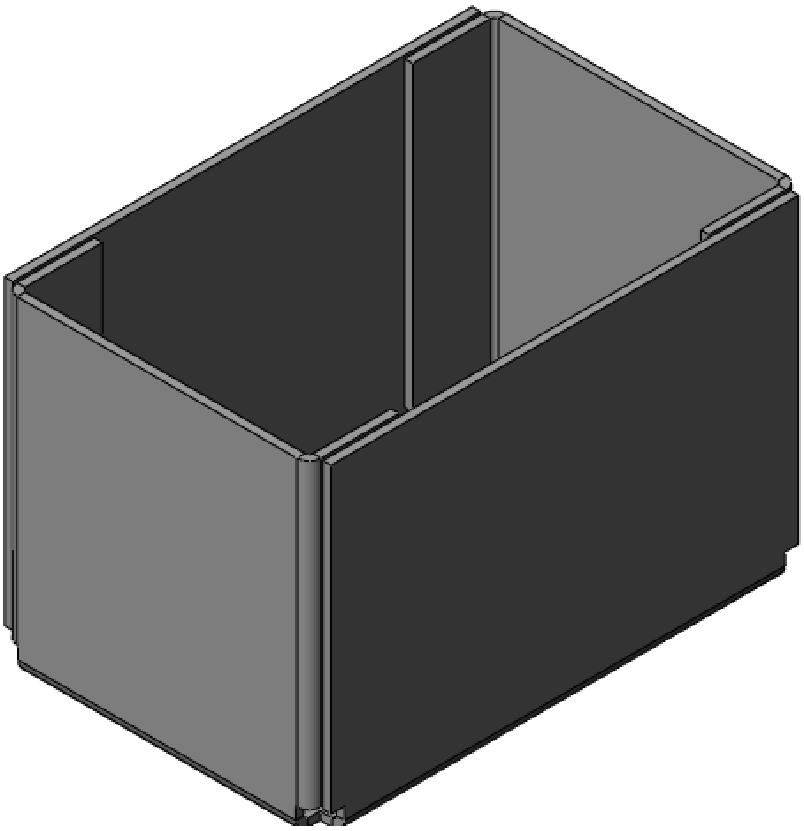
    For more information about adding a bill of materials, see bill of materials in the **SolidWorks Online User's Guide**.

Chapter 5:
Sheet Metal

This chapter covers the concept Sheet Metal Part. You will learn to create a bended Sheet Metal Part and different features to edit the Part.

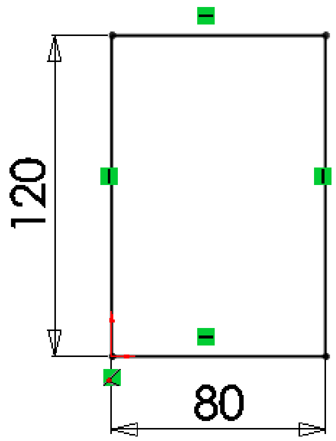
Creating a Sheet Metal Part

When developing a sheet metal part, there are two ways to model it. One way is to create a normal solid part, then turn it into a sheet metal part. This part of the manual explains the basics of the Sheet Metal Part. You will learn to create the model below.



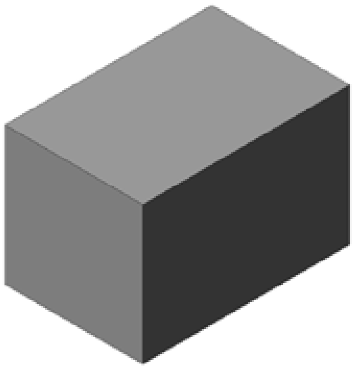
Extruding a Block

You can create a sheet metal part based on any part with a uniform thickness. One way to create this type of part is to extrude a block and then shell it.



1. Open a new part file, and open a sketch on the **top** Plane.

2. Sketch and dimension the lines as shown. Make sure the lower-left corner is **coincident** with the **Origin**.



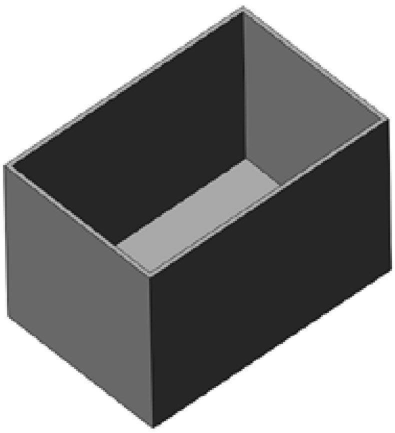
3. Click **Extruded Boss/Base** or **Insert, Boss/Base, Extrude...**

4. Under **Direction 1**:
   - Set **End Condition** to **Blind**.
   - Set **Depth** to **80 mm**.

5. Click **OK**.

Shelling the Part

Now that we have created a block, we can shell it.



6. Click **Shell** or **insert, Feature, Shell...** The **shell** PropertyManager appears.

7. Select the **Top** face for **faces to remove**.

8. Set the **Thickness** to **2 mm**.

   In the next part we will start using Sheet Metal features. It is possible to have a Tab with sheet Metal Features next to Features and Sketch in the CommandManager.

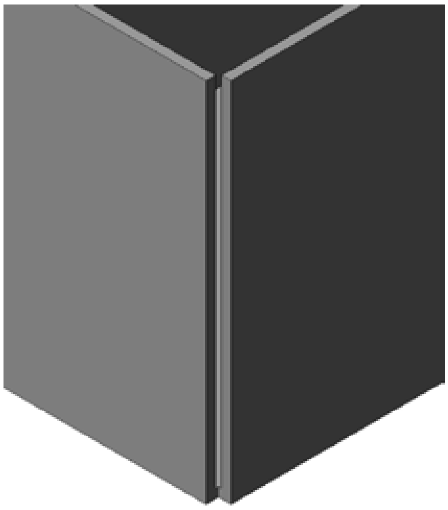
9. Right-click one of the tabs of the **CommandManager** and select **Sheet Metal**.

   In the next part we will start using Sheet Metal features. It is possible to have a Tab with sheet Metal Features next to Features and Sketch.

10. **Right**-click one of the tabs of the CommandManager and select **Sheet Metal**. The Sheet Metal tab appears next to Features and Sketch.

Ripping the edges

A Sheet Metal Part exists out of one sheet of metal, that is cut and then bent into the desired shape. As you can see, the Part above Part cannot be created out of onesheet of Metal. We need to open up the edges between the walls. This can be achieved using the **Rip** Feature.



11. Click **Rip** on the **Sheet Metal** tab or **Insert, Sheet Metal, Rip...** The **Rip** PropertyManager appears.

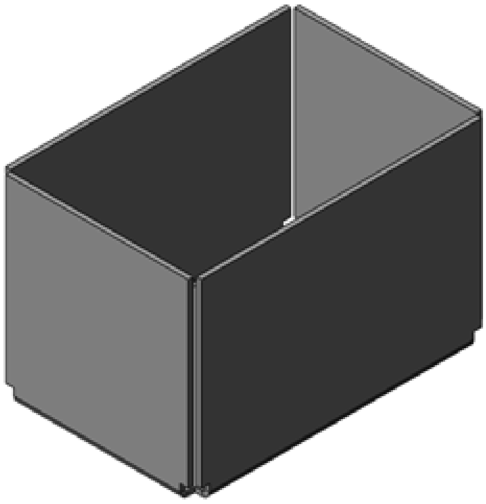
12. Under **Rip Parameters**:
    - Select the **four inner vertical edges** as **Edges to Rip** .
    - Set the **Rip Gap** to **1 mm**.

13. Make sure that you see two arrows in each corner. This way the 1 mm that will be cut away will be equally divided between the two edges.

14. Click **OK**.

Inserting Sheet Metal Bends

Now that the walls are separated, it is possible to convert the extruded base to a sheet metal part. SolidWorks can find edges that can be transformed into bends. You can specify the fixed face, bend radius and bend allowance. The software creates the bend lines.



15. Select the **bottom** face inside the part. This face will remain fixed when the part is bent or flattened.

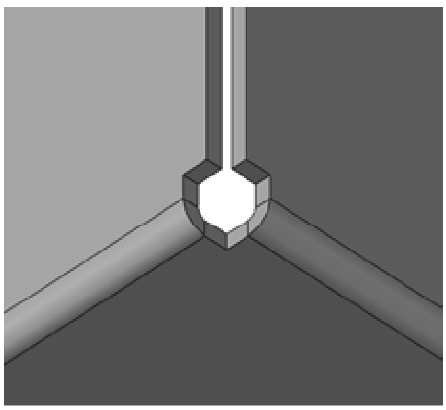
16. Click **Insert Bends** on the **Sheet Metal** tab, or **Insert, Sheet Metal, Bends**. The **Insert Bends** PropertyManager appears.

17. Under **Bend Parameters** specify a **Bend Radius** of **1 mm**.

18. Under **Bend Allowance**, make sure that the **Bend Allowance Type** is set to **K-Factor**, with a value of **0.5** .

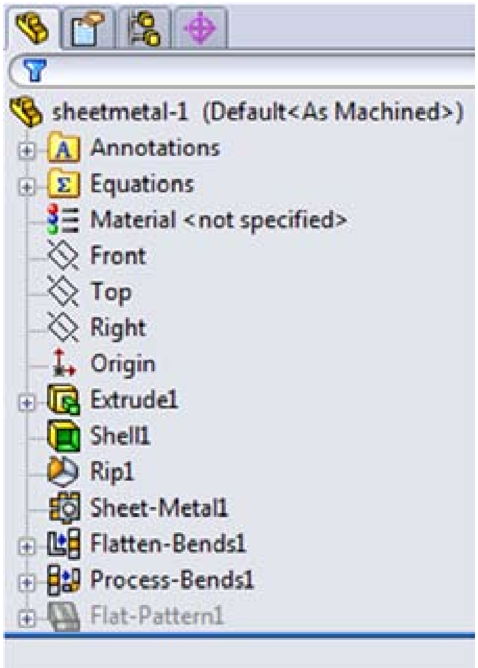
19. Make sure that **Auto Relief** is checked with the **Rectangular** option, with a **Ratio** of **1.5**. This allows the software to add relief cuts wherever necessary in order to make the bends. **Rectangular reliefs** are square cuts of the size required to insert the bend and flatten the Part.

20. Click **OK**. The sharp corners of the base are converted to curved bends.



Rolling Back the Design to the Flattened State

Examine the FeatureManager design tree. There are four new features listed, representing the steps in the process of creating a sheet metal part.



- **Sheet-Metal**
  the Sheet-Metal feature marks the beginning of the process. It contains the default bend parameters.
- **Flatten-Bends**
  The Flatten-Bends feature converts the bent-up base into a flat sheet, with bend-lines in the necessary places.
- **Process-Bends**
  The Process-Bends feature folds (processes) the flattened part, returning it to its bent-up state.
- **Flat-Pattern**
  The Flat-Pattern feature is intended to be the last feature in the folded sheet metal part. All features before Flat-Pattern in the FeatureManager design tree appear in both the folded and flattened sheet metal part. All features after Flat-Pattern appear only in the flattened sheet metal part. The Flat-Pattern feature is suppressed by default. The feature is mostly used for drawings and to check if it is possible to create the shape out of one Sheet of Metal.

When you add new features to the model, they are automatically added just above **Flat-Pattern**.

Unfolding the Sheet Metal Part

When using Sheet Metal, you can unfold and (re)fold the Sheet Metal. Sometimes this is very useful when editing difficult or not visible faces. There are four folding features: **Unfold** , **Fold** , **Flatten** and **No Bends** .

With **Unfold** you can flatten one or more bends. After editing the flattened Part, you can use **Fold** to put the Part back in its original state.

Flatten is used to make the entire Part flat. This is used for example when making Drawings. Another way to do this is by **right**-clicking **Flat-Pattern** in the FeatureManager and select **Unsuppress**.

Finally there is the **No Bends** feature. This feature temporarily puts the Sheet Metal Part back in its state just before the bends were inserted. The same can be achieved by dragging the rollback bar above **Flatten-Bends** or by **right**-clicking **Flatten-Bends** in the FeatureManager and select **rollback** .

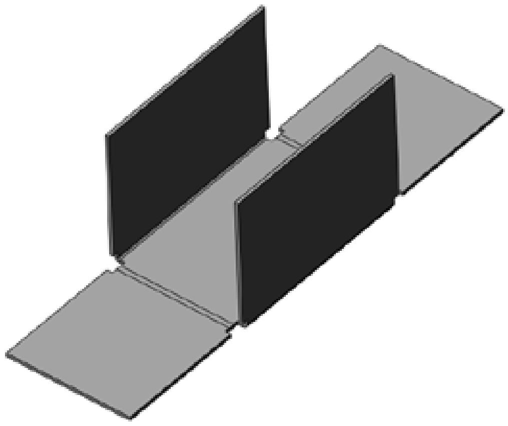
In our Part we want to connect the four walls with each other and thus close the gaps in the corners. We are going to do this by creating **Miter Flanges** on the edges of the **front** and the **back** wall, which can be point-welded to the side walls later on in production. To do this in an easy way, we first need to **Flatten** the **front** and the **back** wall.

21. Click **Unfold** or **insert, Sheet Metal, Unfold...** The **Unfold** PropertyManager appears.

22. Select the bottom face inside the box as the **fixed face** . This face will stay the same and the other faces will be unfolded with regard to this face.

23. Select the bends of the short walls as **Bends to Unfold** .

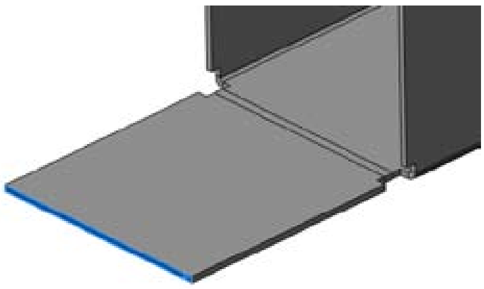
24. Click **OK**.



Adding a Miter Flange

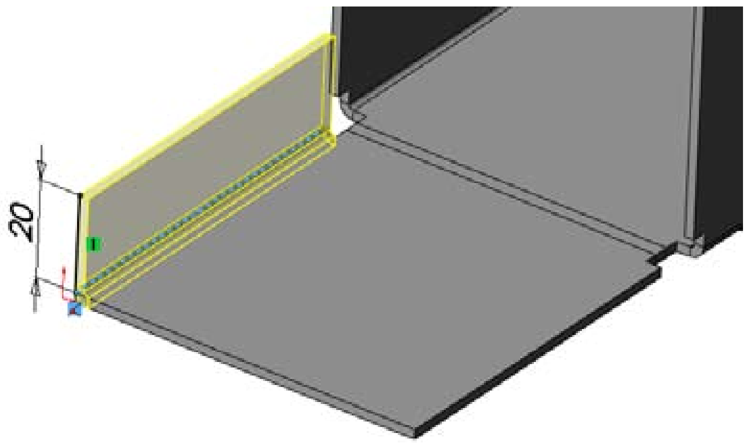
As told earlier, We will add Miter Flanges to the Part. These are added using the **Miter Flange** feature.

25. First, **open** a sketch on the **outer** face of the front wall, as shown below.



26. Draw a **vertical** line from the **top left** corner. Dimension it **20 mm**.

27. On the Sheet Metal tab, Click **Miter Flange** or **Insert, Sheet Metal, Miter Flange...** The **Miter Flange** PropertyManager appears.



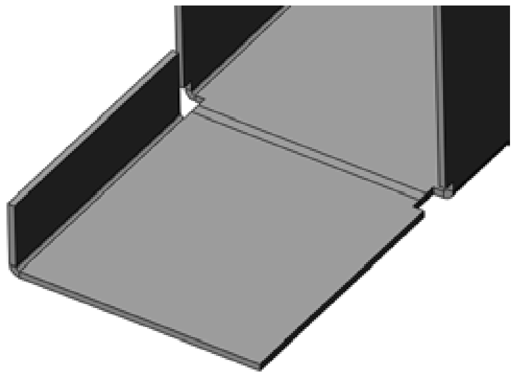
SolidWorks Also shows a preview of the Miter Flange. As you can see, the edge perpendicular to our sketch is automatically added to **Along Edges** .

An important option here is **Flange position**. You can select three options:
Material Inside , **Material Outside** and **Bend Outside** . Sometimes the outer dimensions needs to stay the same. So the bends must be within the dimensions given to the outside of the part. In this case, you can choose **Material Inside**. If the inside dimension needs to stay the same, like when something needs to fit inside the Part, you can choose **Material Outside**. That way, the Flange is exactly outside of the original dimension. Sometimes the bend needs to be completely outside of the original measurement to prevent a relief cut. In this case, **Bend Outside** is the right choice.

Trim Side Bends is also an option. This option is used to cut away material from a tangent corner, if necessary. It gives a corner a better finish, without relief cuts.

Another option is the **Gap distance**. It is the gap between two bends. This option is used to set the gap between two Miter Flanges when a Miter Flange feature is applied to more than one edge. On the one hand it is preferable to keep it as small as possible, on the other hand, you have to keep the tolerances in mind. Usually the Gap Distance has the same value as the Bend Allowance.

28. Under **Miter Parameters**:
    - Make sure **Use Default Radius** is checked.
    - Set **Flange Position** to **Material Inside** (try the different options and see the difference).
    - Check **Trim Side Bends**.
    - Since the Miter Flange doesn't continue on another edge, the **Gap Distance** isn't very relevant. You can leave it at its original value.



29. Click **OK**.

Mirroring the Miter Flange

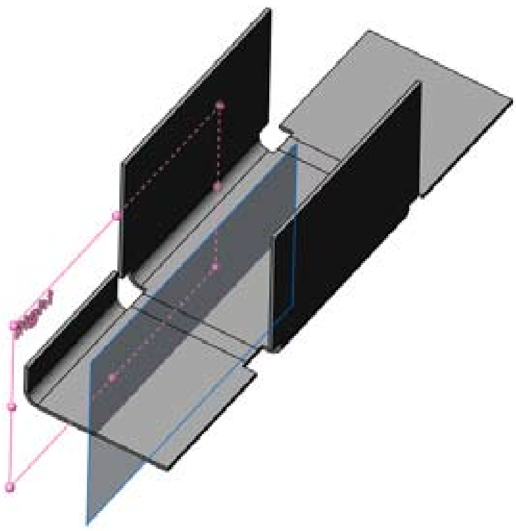
After you created the Miter Flange, we need to mirror it to get it on all the four edges of the flattened walls. We start off by creating two **Planes**. These two planes are both placed at **half** the **length** and the **width** of the Part.

30. Select the **Right** Plane.

31. On the Features tab, Click **Reference Geometry** and **Plane** or **Insert, Reference Geometry, Plane...** The **Plane** PropertyManager appears.

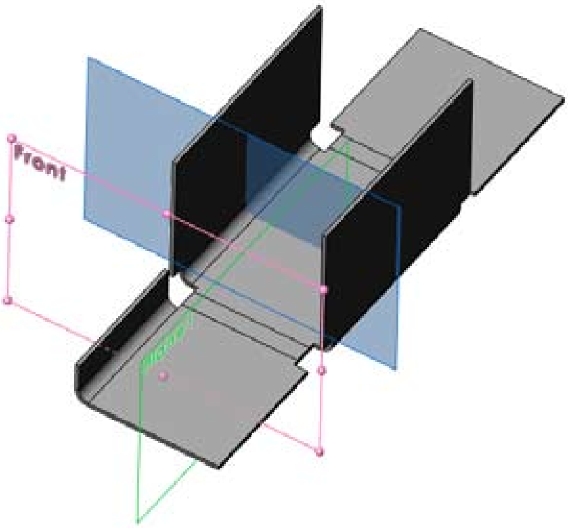
32. Select **Offset Distance** and set it to **40 mm**.

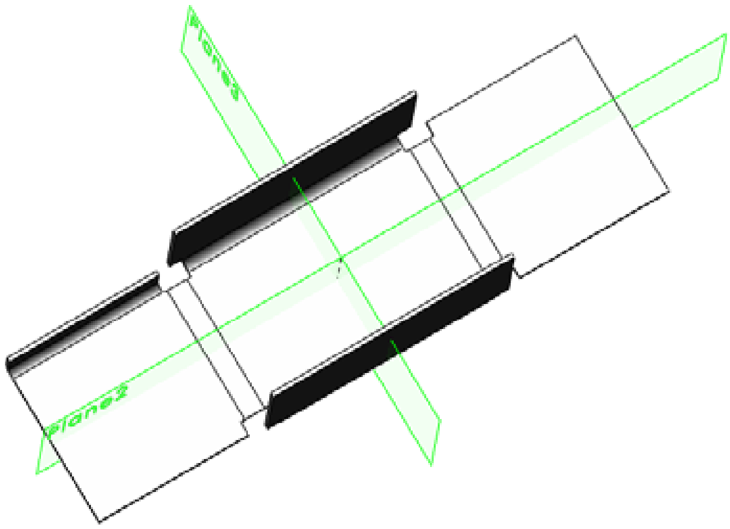
33. If necessary, check **Flip** to change the direction to be the same as shown to the right. Click **OK**.



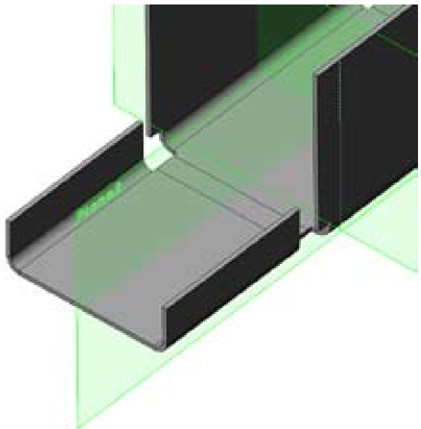
34. Select the **Front** Plane.

35. Create a second **Plane** with an **Offset Distance** of **60 mm**. **Flip** it to place it in the middle of the Part, as shown below to the left.





Now that both planes are in place we will **Mirror** the Miter Flange to all four edges.

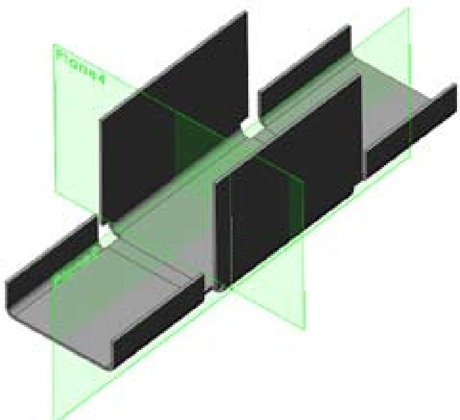


36. Click **Mirror**  , or **Insert, Feature, Mirror...** The **Mirror** PropertyManager appears.

37. Select the **first Plane** as the **Mirror Plane**.

38. Select the **Miter Flange** as **Feature to Mirror**.

39. Click **OK**. The result should be the same as shown to the right.



40. Select **Mirror** again.

41. Select the **second Plane** as the **Mirror Plane**.

42. Select both **Miter Flanges** as **Features to Mirror**.

43. Click **OK**.

Now we have Miter Flanges on all four edges.

Folding the Front and Back wall.

Now that we created the Miter Flanges, the walls can be bent upwards again. As explained earlier, we will use the **Fold** Feature for this.

44. Click **Fold** or **Insert, Sheet Metal, Fold…** The **Fold** PropertyManager appears.

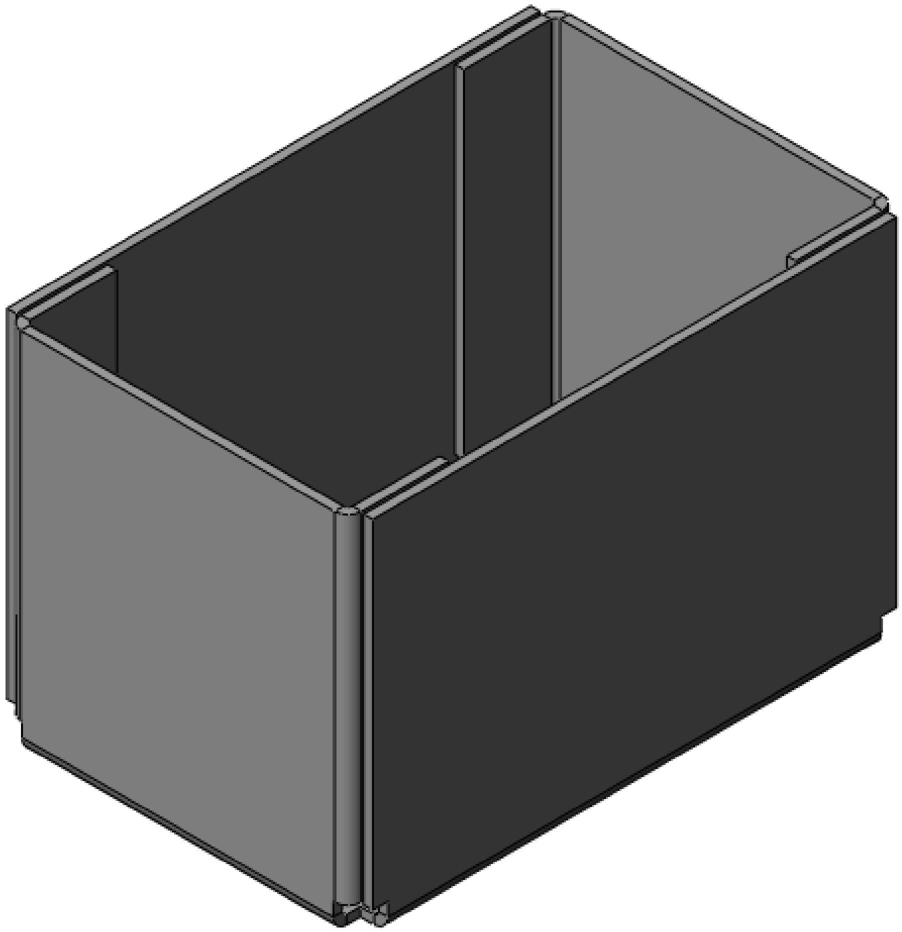
45. Select the **bottom** face again as the **Fixed Face** .

46. Click the **Collect All Bends** button. Solidworks will select all Unfolded bends.

47. Click **OK**.

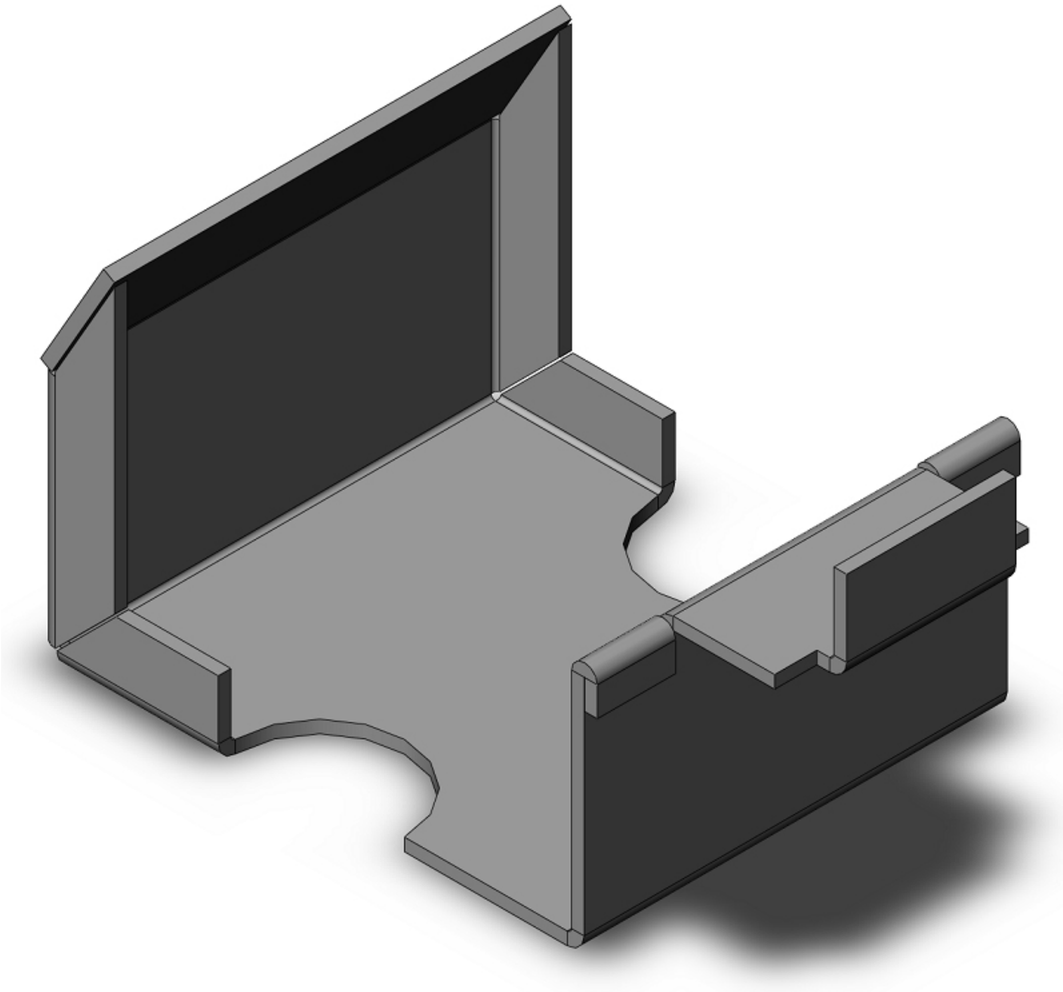
48. Save the part as **‘sheetmetal1.SLDPRT’**.

The Part is ready now.



Other features for Sheet Metal

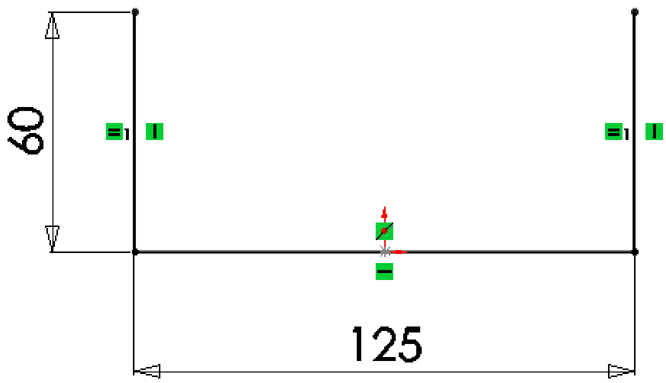
In the previous example, you have learned how to create Sheet Metal starting with a solid model. This part of the manual shows how to create Sheet Metal with the Base-Flange Feature. Furthermore, you will learn how to use some extra Sheet Metal features that are useful when modeling Sheet Metal.

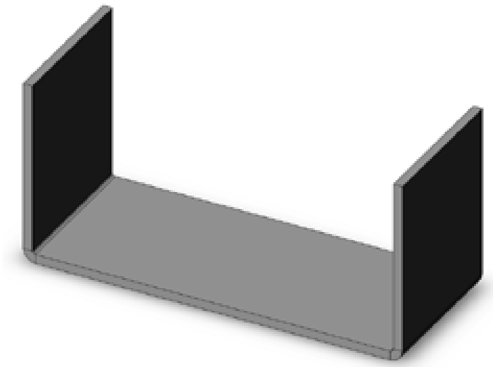


Creating a sheet metal Part using Base-Flange

1. Make a new part file, and open a sketch on the **front** Plane.

2. Sketch and dimension the lines as shown. Make sure it is fully defined; the two vertical lines have an **equal** relation and the **origin** is the **midpoint** of the horizontal line.





3. Click **Base-Flange** or **Insert, Sheet Metal, Base Flange...**

4. Under **Direction 1**, Set the **Depth** to **50 mm**.

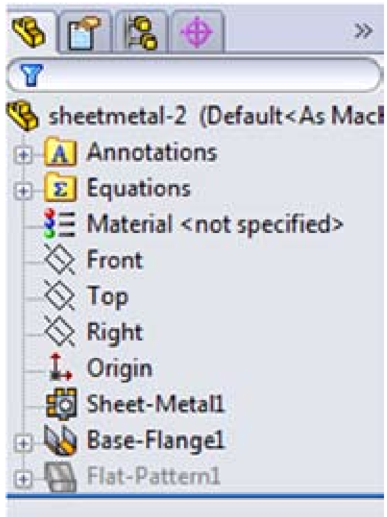
5. Under **Sheet Metal Parameters:**
   - Set **Thickness** to **3 mm**.
   - specify a **Bend Radius** of **1 mm**.

6. Under **Bend Allowance**, make sure that the **Bend Allowance Type** is set to **K-Factor**, with a value of **0.5**.

7. For **Auto Relief**, select **Rectangular** in the drop-down list, check **Use relief ratio** and set the **Ratio** to **0.5**.

8. Click **OK**.

   As you can see, the Design Tree is somewhat different from last time. Instead of an **Extrude**, possibly a **Rip**, **Flatten-Bends** and **Process-Bends**, it now contains **Base-Flange**. Base-Flange now contains the first Solid feature of the Sheet Metal part and the first bends.

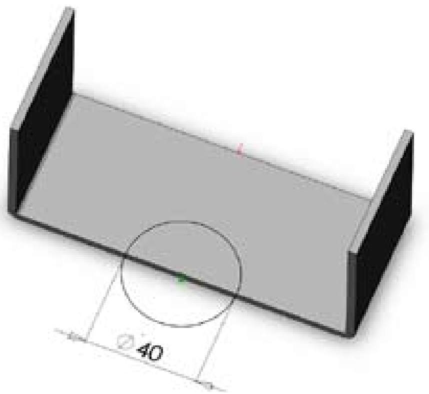


Adding a hole to the Part

Next, we will add a hole to the Part.

9. Open a new sketch on the bottom face

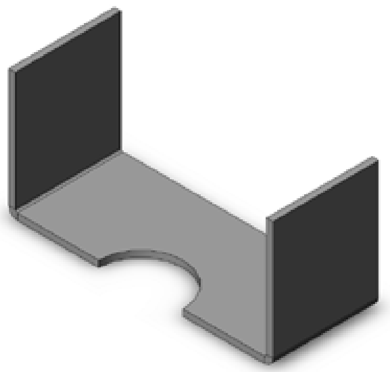
10. Draw a circle and dimension it as shown. Make sure the center is the **midpoint** on the edge.



11. Create an **Extruded Cut** or **Insert, Feature, Extruded Cut**.

12. Under **Direction 1**, set the **End condition** to **Through All**.

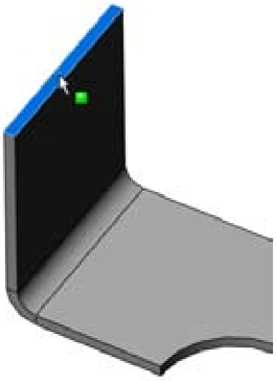
13. Click **OK**.

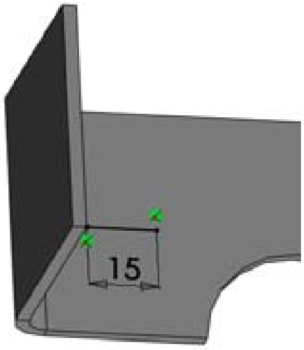


Adding a Miter Flange

Next, we want to add a Miter Flange.

14. Select the top face of the left wall and open a Sketch.





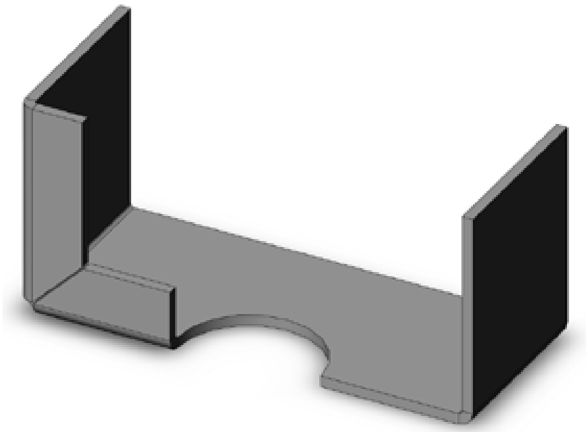
15. draw a horizontal line and dimension is **15 mm**.

16. While still in the Sketch, Click **Miter Flange** or **Insert, Sheet Metal, Miter Flange...** The **Miter Flange** PropertyManager appears.

17. On the edge you can click on **Propagate** . This way the Miter Flange will continue on the tangent edges. These edges are also added to **Along Edges** . As you can see, the Flange stops at the notch we created earlier.

18. Under **Miter Parameters**:
    - Set **Flange Position** to **Bend Outside** .
    - Set the **Gap Distance** to **0.50 mm**. this time, the Gap Distance does matter.

19. Click **OK**.

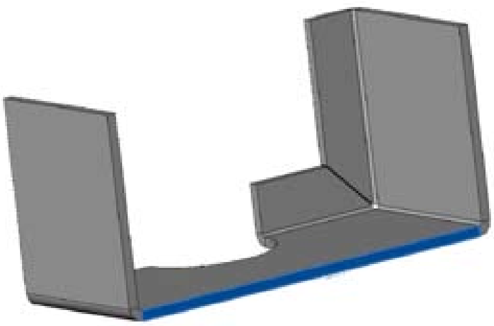


Mirroring the body

As you might have noticed, the Sheet Metal Part we created is only half the size of the first image. That is because the Part is symmetrical, so we can use the Mirror feature half the Part. Now we will use the Mirror feature to create the full part.

20. Click **Mirror** or **Insert, Pattern/Mirror, Mirror...**

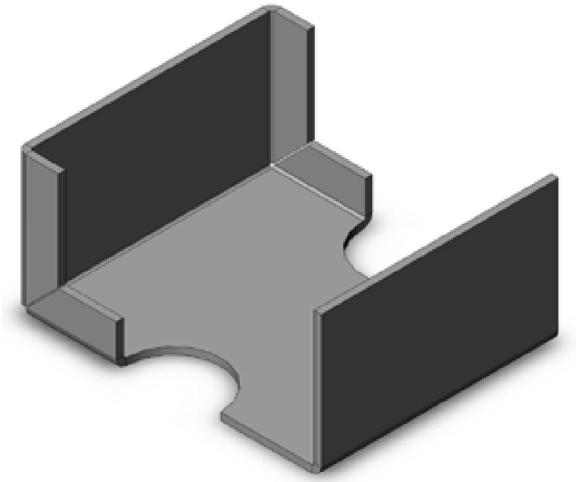
21. Select any face on the back as **Mirror Plane**.



22. Expand **Bodies to Mirror**, and click anywhere on the part to select the complete body to mirror.

23. Click **OK**.

    The Part should now look like this:

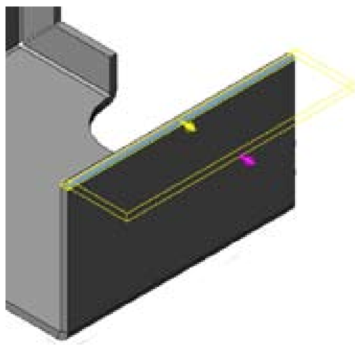


Creating an Edge Flange

If you want to add an extra edge to your Sheet Metal Part, you can use the **Edge Flange** feature.

24. Select the top edge of the right wall.

    Note: *It doesn't matter which edge of the top face you select.*



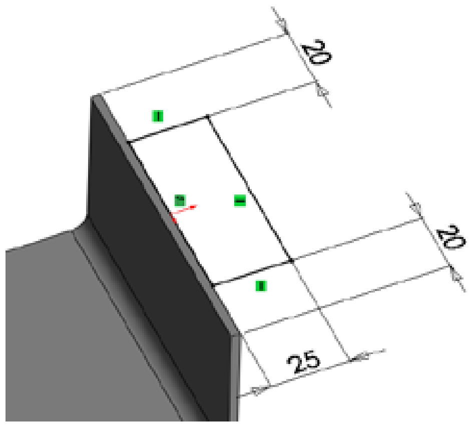
25. Click **Edge Flange** or **Insert, Sheet Metal, Edge Flange...**

26. Pull the Flange to the right to make it stand outward.

27. Under **Flange Length**:
    - Set **Length** to **25 mm**.
    - Select **Inner Virtual Sharp** .

    This way the Flange sticks out exactly 25 mm. With **Outer Virtual Sharp** selected, the Sheet Metal Thickness of the already existing wall is taken into account in the length.

28. Under **Flange Position**, set the **Flange position** to **Material Inside** .



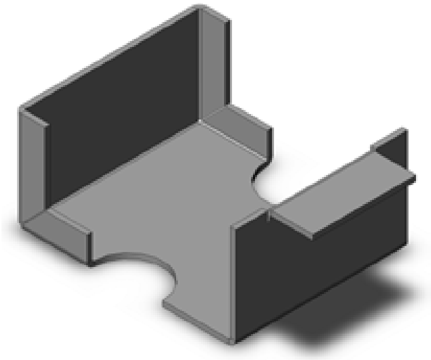
    We don't want to create a Flange along the whole edge. We will edit the it's shape using **Edit Flange Profile**.

29. Click **Edit Flange Profile**. A **profile Sketch** dialog appears. Move this dialog to the side to view and edit the sketch created by the feature.

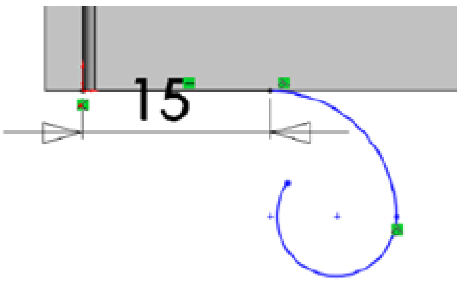
30. Move both short sides from the edge endpoints. Dimension the rectangle as shown.

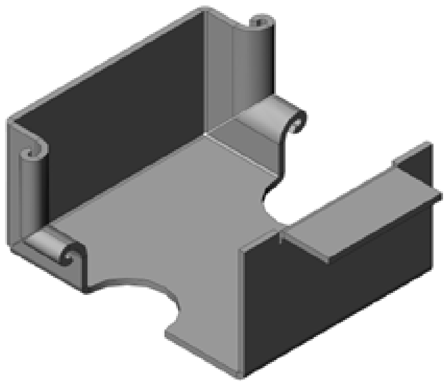
31. In the **Profile Sketch** dialog, click '**Back**' (or '**Vorige**', depending on your system language). The propertyManager re-appears, and a preview is shown.

32. Click **OK** to finish the Edge Flange.



You might wonder what the difference is between the **Miter Flange** and the **Edge Flange** feature. With Edge Flange, you can edit the shape of the surface and the angle, but it will always be on one plane. With Miter Flange you can't specifically edit the shape of the surface, but the Flange doesn't have to be on a plane, with one specific angle with regard to the edge on which it is created. In the example below, you can see an edited version of the Miter Flange. It is clear that the Flange is no longer on one plane (this is just an example, the tutorial will continue without it).





Creating a Hem

As you can see, the edge on the right is now a strange edge that is still straight on the sides. We can solve this by creating two hems. By creating a Hem we curl an edge, giving it a smoother finish.

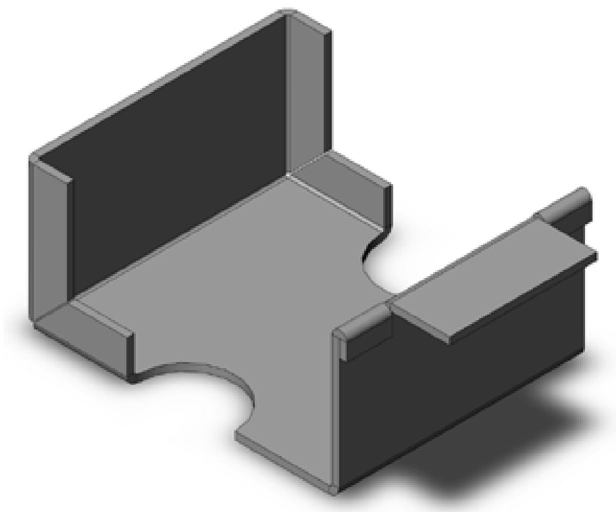
33. Select both edges, using the **Ctrl**-key.

34. Click **Hem** or **Insert, Sheet Metal, Hem...**

35. Under **Edges**:
    - Select **Reverse Direction** , to curl the Hem to the outside, if needed.
    - Select **Material Inside** .

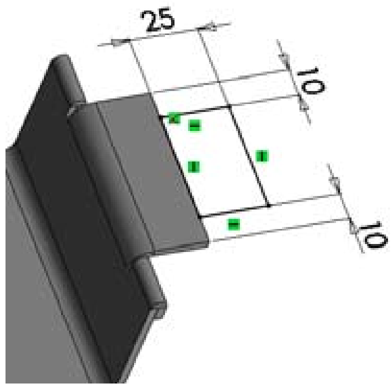
36. Under **Type and Size**:
    - Set **Type** to **Closed** .
    - Set **Size** to **10 mm.**

37. Click **OK**.



Adding a Tab

If you want to add material to a face of your Sheet Metal Part, you can do this with the **Tab** feature.



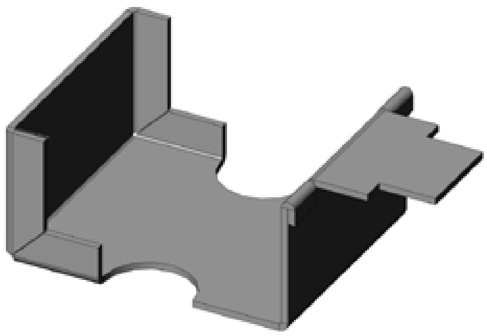
38. Open a new sketch on the Flange on the right wall (the one you created earlier with the Edge Flange feature).

39. Draw a rectangle on the edge and dimension as shown.

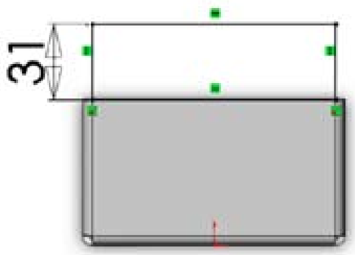
    Note: the difference between ***Edge Flange*** *and* ***Tab*** *is that Edge Flange will add a bend, while Tab can be used to extend any surface.*

40. Click **Tab**  of **Insert, Sheet Metal, Tab...**

41. After the PropertyManager appears, click **OK**. The tab is added.



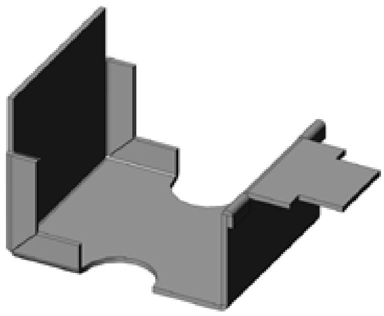
On the left wall, we also want to create an extra Tab.



42. Open a sketch on the outside face of the left wall.

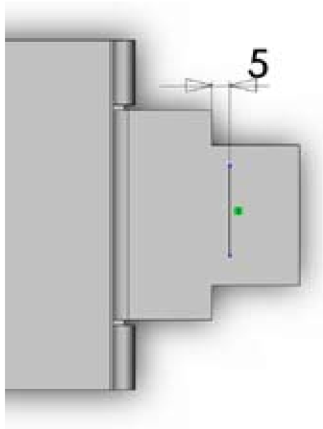
43. Sketch a rectangle **coincident** with the edge of the face, with a height of **31 mm** and with the same width as the face.

44. Click **Tab**  or **Insert, Sheet metal, Tab...** The Part should now look like the example shown to the right.



Bending a Tab

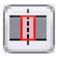
The tabs we just created need to be bent. You can bend a wall by sketching a bend-line and use the Sketched Bend feature to bend the wall at the bend-line and at the desired angle



45. Open a new sketch on the first Tab we just added.

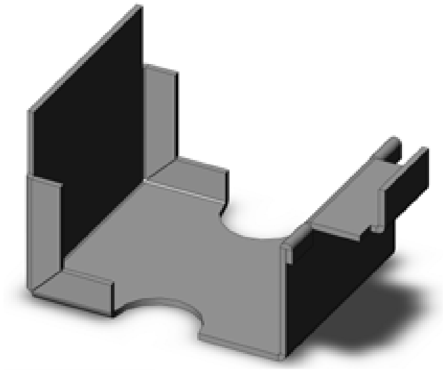
46. Draw a line and dimension as shown. Make sure the line is vertical. Notice that the line doesn’t need to be the exact same length of the face you are bending.

47. Click **Sketched Bend**  or **Insert, Sheet Metal, Sketched Bend...**

48. Under **Bend Parameters**:
    - Select the face we are bending on the left side of the sketched line as the **Fixed Face**  .
    - Set **Bend Centerline**  as **Bend position**.

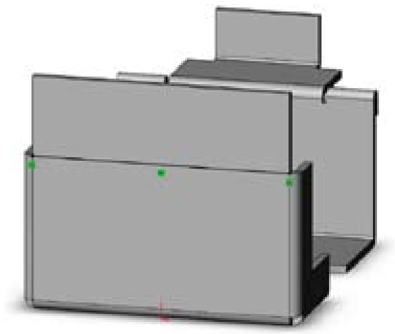
    You will notice a black dot appearing on the face. That side of the bending line will stay fixed. If you select the face on the right, then that face will stay fixed and the whole Sheet Metal Part will be bent around it, which of course isn’t what we want here.

    - We want a **90°** bend, so you can leave the **bend Angle** as it is.



49. Click **OK**. The Tab now has an upward bend.

The second Tab needs to be bent **30°** inward.

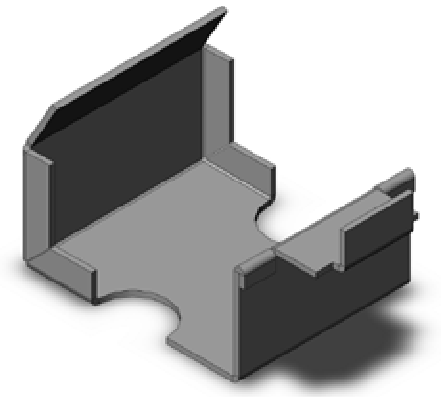


50. Open a new sketch on the outside face of the left wall.

51. Draw a line as shown. The line needs to be **coincident** with the two corner **vertices**, to make sure it’s exactly on that line.

52. Click **Sketched Bend**  or **Insert, Sheet Metal, Sketched Bend...**

53. Under **Bend Parameters**:
    - Select the face we are bending below the sketched line as the **Fixed Face**  .
    - Set **Bend Outside**  as **Bend position**.
    - Set the **Bend Angle** to **30°** .
    - The face needs to be bent inward, so check **Reverse Direction**  .



54. Click **OK**.

Now you have bent both Tabs.

Creating a closed corner

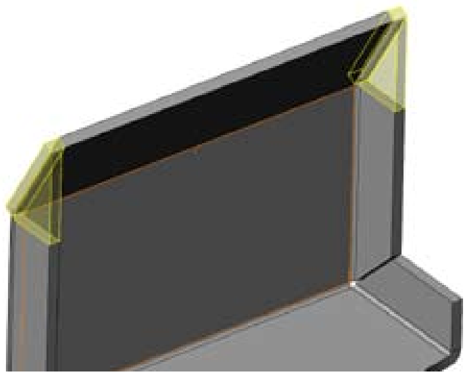
As you can see, there is a gap on the left side between the Miter Flange and the Tab we just bent. We can fill it up using the **Closed Corner** feature.

55. Click **Corners** **, Closed Corncer** or **Insert, Sheet Metal, Closed Corner...**

56. Select the top faces of the Miter Flanges as the **Flanges to Extend** and the sides of the bent tab as **Faces to Match**.

The second option you see is the **Corner Type**. Here you can choose to make the gap between the two faces equal, or let one of the two overlap the other.

In our case we want the Tab overlap the Miter Flanges. Because we selected the top faces of the two Miter Flanges, we need to choose **Underlap**.



57. Set **Underlap** as the **Corner Type**.

58. Set the **Gap Distance** to **0.5 mm**.

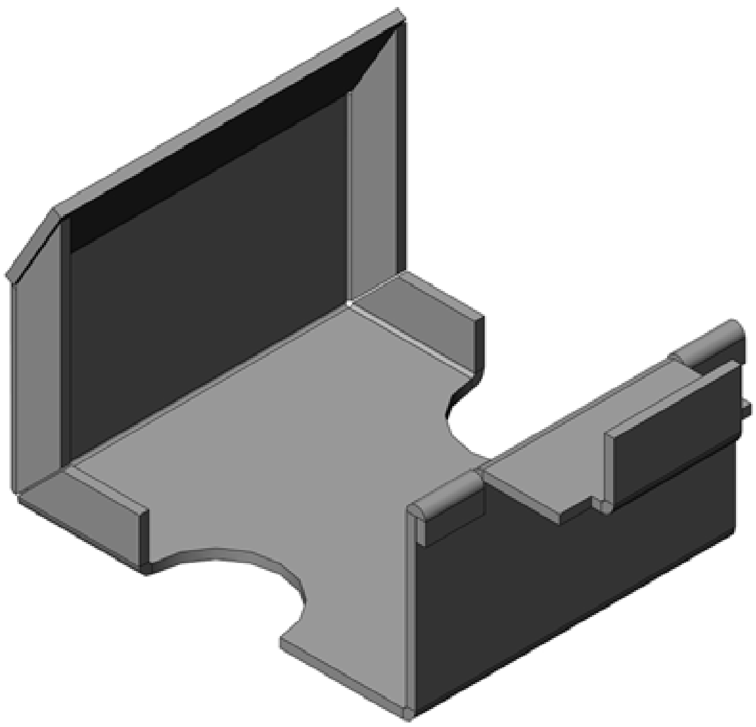
59. Check the **Open Bend Region** option.

60. Click **OK**.

Now the gaps are filled.

61. Save the part as **sheetmetal-2.SLDPRT**.

The Sheet Metal Part is done.



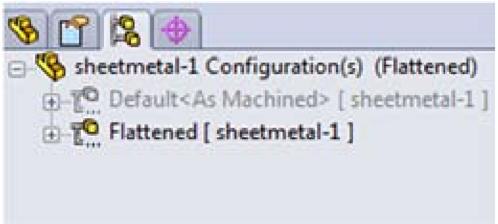
Creating Drawings for Sheet Metal Parts

Saving different configurations

It can be useful save two different configurations of a Sheet Metal Part, One of the bended part and one of the flattened part.

1. Switch to the **ConfigurationManager** tab .

2. Right-click the part icon and choose **Add Configuration** from the menu.



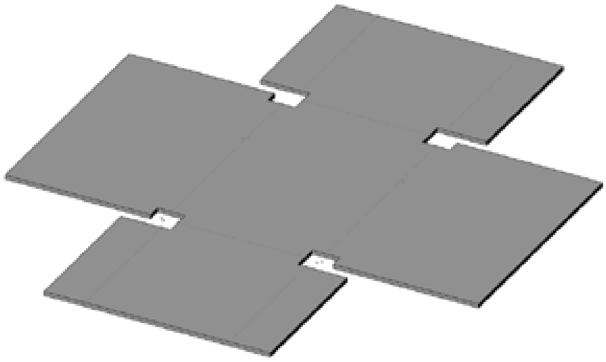
3. Type in **Flattened** as Configuration Name and click **OK**.

   A new configuration is added in the ConfigurationManager and is made the active configuration.

4. Return to the FeatureManager.

5. Click **Flatten** in the **Sheet Metal** Tab of **Right**-click **Flat-Pattern** and choose **Unsuppress** from the menu.

6. Save the part.



When saving the part, the flattened configuration is assigned to the added configuration. When double clicking the configuration **Default** in the ConfigurationManager, the part will return to its bended state.

Creating Drawings of Sheet Metal Parts

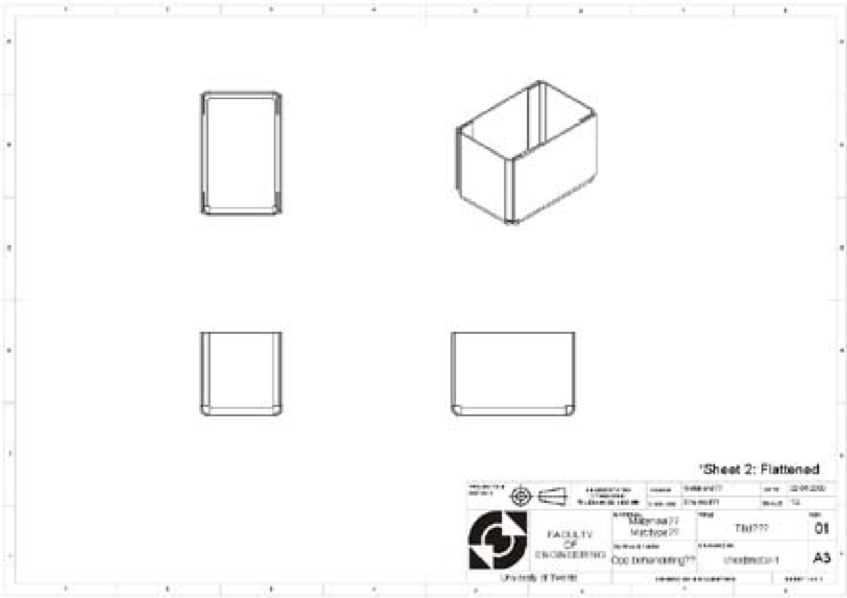
When working with sheet metal parts, it is often necessary to create a drawing sheet that shows the part in its flat or unbent state. This is accomplished using configurations and the **Flat-Pattern** feature. But first, we will make a standard drawing of the part, as shown on the next page.

7. Open a new drawing with a **drawing_ut_a3_hor** template and name the sheet **'Bended'**. If not already, set the sheet scale to **1:2**.

8. Add a **Standard 3 View** of **sheetmetal-1.SLDPRT** in the **Default** configuration.

9. Add a **Named View** of the **isometric** view in the **Default** configuration.

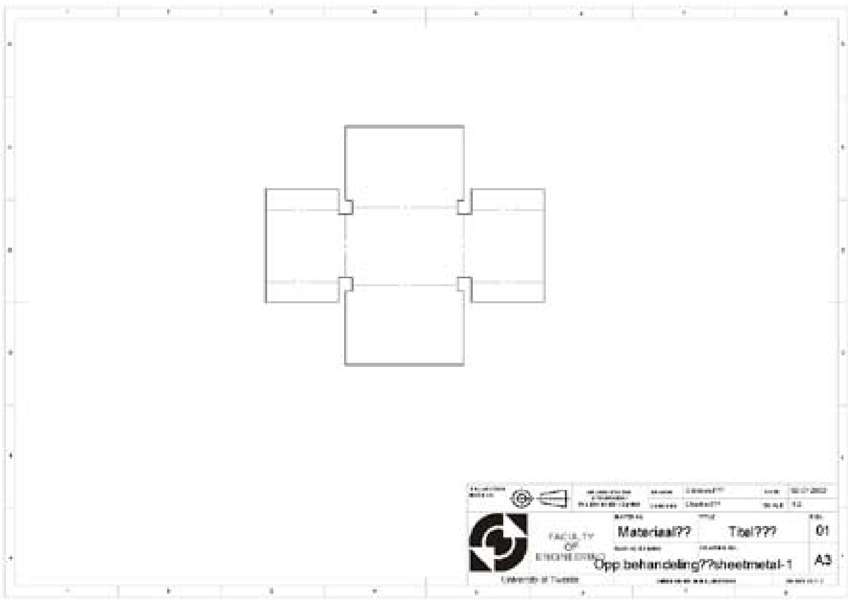
10. Add a comment just above the title block: ***Sheet 2: flattened**.



11. Add a drawing sheet named '**Flattened**'.

12. Add a **Named View** with a **Flat pattern** view orientation. You can find Flat Pattern under **More Views**.

13. SolidWorks automatically adds all the bend annotations. But we don't need these this time. Select them all, right-click and choose **Hide**.



The drawings are now done.

14. Save the Drawings.

Creating a dxf-file for Sheet Metal Production

For difficult forms it may be easier of cheaper to, for instance, use laser cutting in the process. For such operations are usually dxf-files used. For this purpose we will create a dxf-file for our 'sheetmetal-2' part.

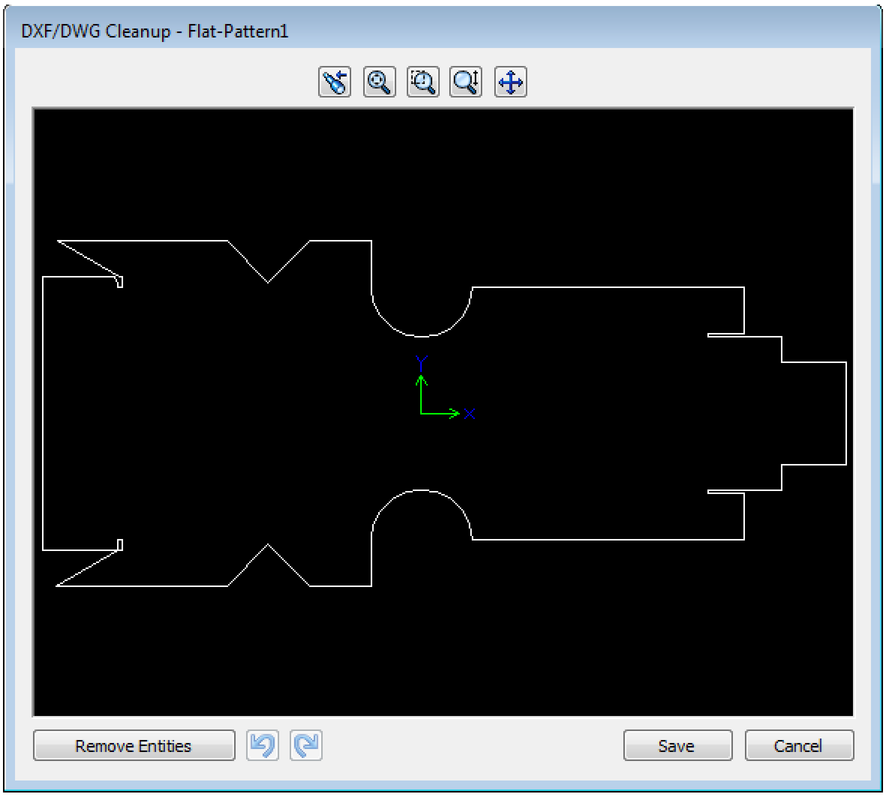
15. Open '**sheetmetal2**' is it is not still opened yet.

16. Select **File**, **Save as...**

17. In the save dialog, select **.dxf** as the filetype. The name is automatically changed to '**Flat Pattern - sheetmetal2.dxf**'. Click **save**. The **DXF/DWG Output** PropertyManager appears.

18. Select the following properties:
    • Under **Export**, select **Sheet Metal**.
    • Under **Entities to Export**, make sure only **Geometry** is selected.

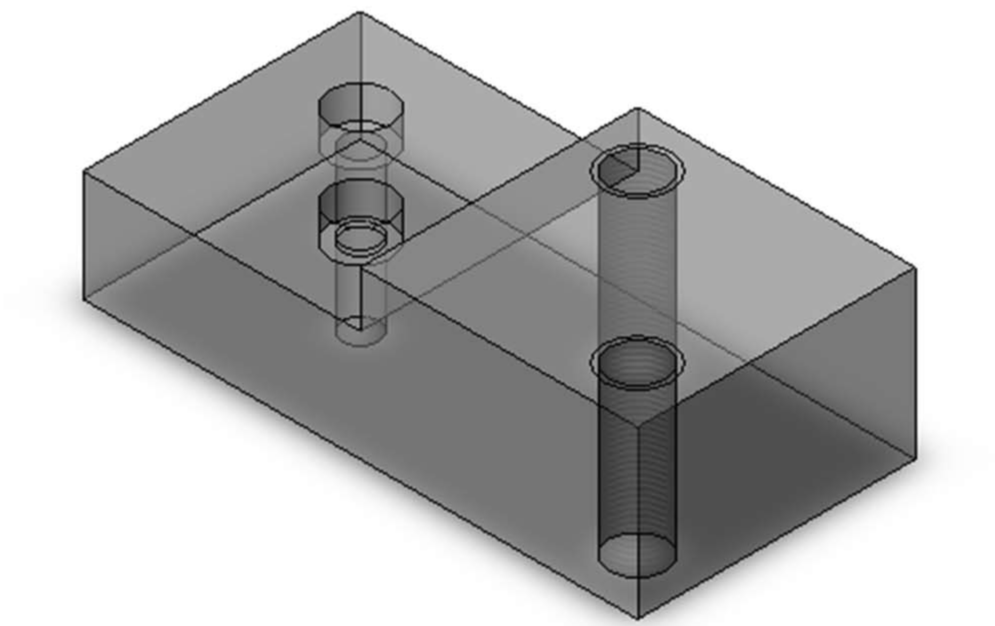
19. Click **OK**. An error dialog appears stating that the default template is nog correct. We don't want to use the sheet format of the standard templates, so select **OK** to continue with a blank sheet.



20. The **DXF Editor** appears. our drawing is already correct, so click **Save**. The dxf drawing is nog ready to be used.

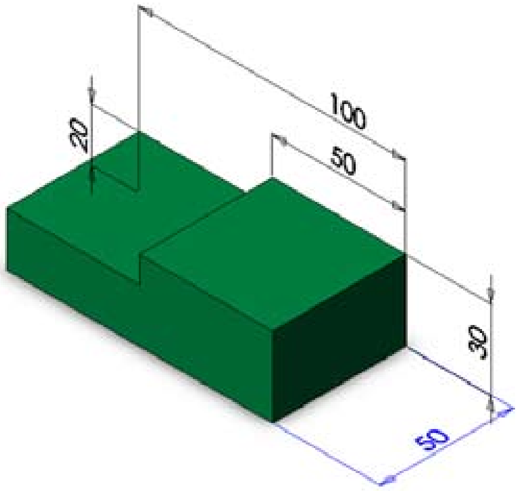
Chapter 6: The Hole Wizard

The Hole Wizard helps you making holes in parts or assemblies. After you have selected the face of the part or assembly, it gives you a number of possible hole types: counterbore, countersink, hole, tap, pipetap, legacy and hole series. The hole type you select determines the capabilities, available selections, and graphic previews. After you select a hole type, you determine the appropriate fastener. The fastener dynamically updates the appropriate parameters. In addition to the dynamic graphic preview based on end condition and depth, graphics in the parameter columns show specific details, as they apply to the type of hole you select. This way the Hole Wizard makes it is easier to create a hole for a specific fastener.

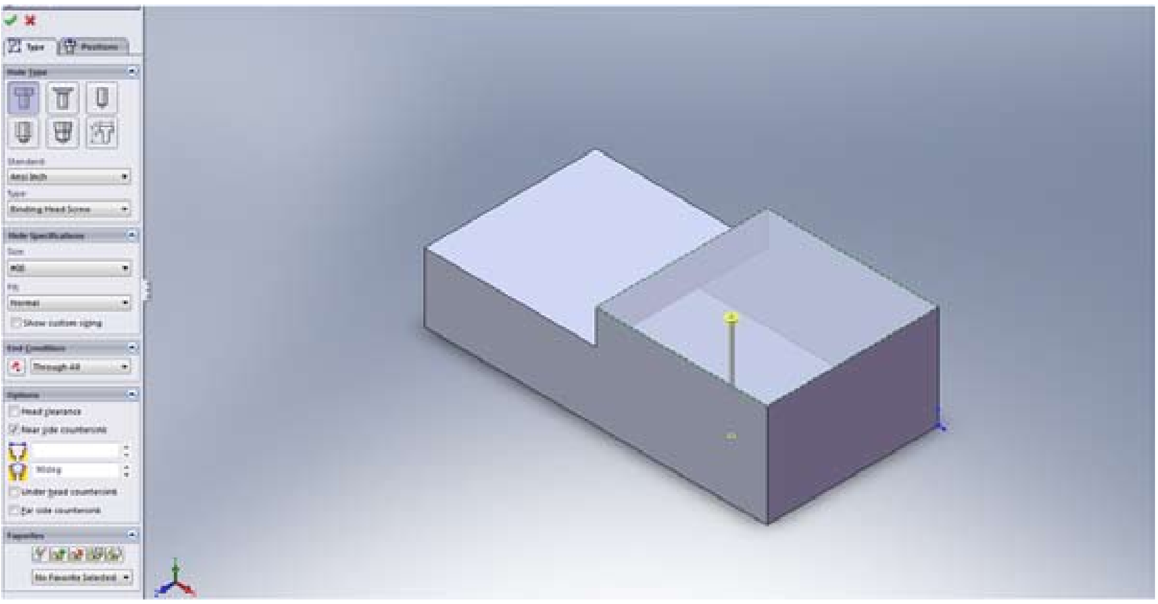


Adding a tapped hole.

1. Create a new part like the picture to the right.



2. Select the **Upper** face, then click **Hole Wizard**  on the **Features** CommandManager or **Insert, Features, Hole, Wizard**.



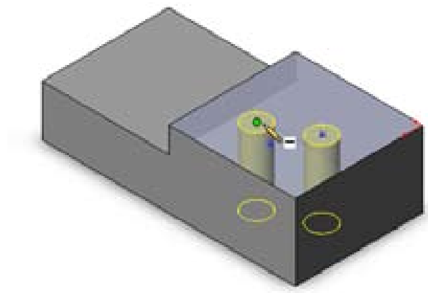
3. The **Hole specification** propertymanager appears. Select the tab **Type**.

   Now you can choose one of the different kinds of holes, like **Counterbore, Countersink, Hole, Tap, Pipetap** and **Legacy Hole**.

4. Select the following options:
   - For **Hole Type** select **Straight Tap**.
   - For **Standard** select **ISO**.
   - For **Type** select **Bottoming Tapped Hole**.
   - For **Size** select **M12**.
   - For **End Condition** select **Trough All**.

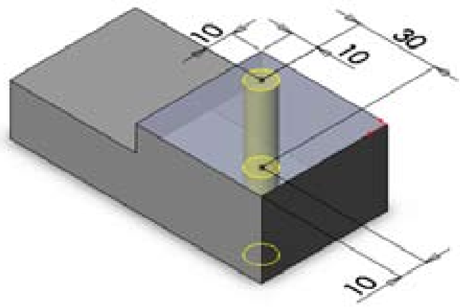
5. Now Select the **Positions** tab.

The Hole Wizard automatically makes the first placement point itself, if you want to create more than one hole, you can place the extra points on the spot where the holes approximately should be.



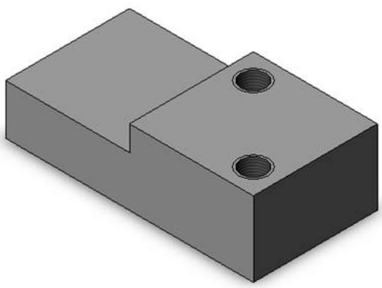
6. Using the **Point** tool, you can add a second point like the picture to the right.

7. Click **Smart Dimension** and add the appropriate dimensions for each hole as given below.



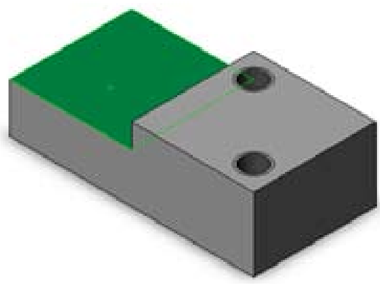
8. Click **OK**.

   The two M12 holes are placed and they appear in de FeatureManager Design Tree.

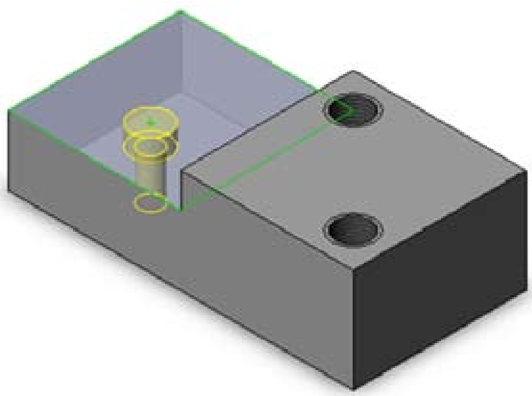


Adding a counterbore hole.

To be able to fix the position of the part to another part, there will be a set of fastening bolts. Now you are going to make the holes for these bolts.



9. Select the **lower** face of the part, then click **Hole Wizard** on the **Features** toolbar or **Insert, Features, Hole, Wizard**.

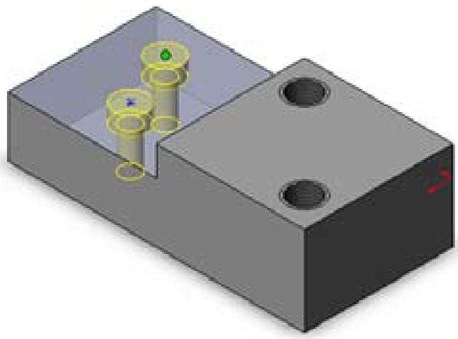


10. The **Hole specification** propertymanager appears. Select the tab **Type**.

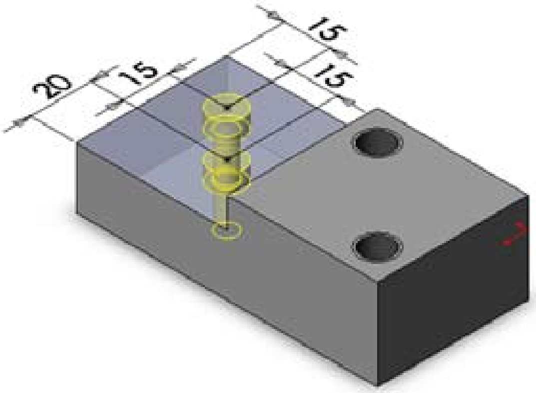
11. Select the following options:
    - For Hole Type select **Counterbore hole**.
    - For **Standard** select **ISO**.
    - For **Type** select **Hex Socket Head ISO 4762**.
    - For **Size** select **M6**.
    - For **Fit** select **Normal**.
    - For **End Condition** select **Trough All**.

12. Select the **Positions** tab.

13. Using the **Point** tool  , add a second point like the picture to the right.

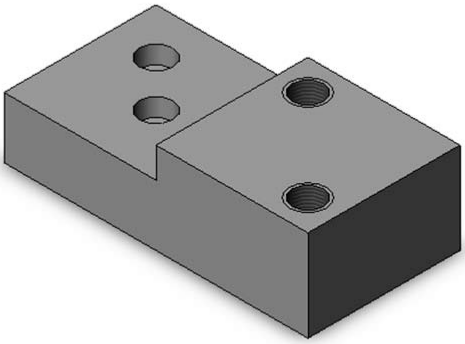


14. Click Smart Dimension  and add the appropriate dimensions for each hole as given below.



15. Click **OK**.

    The two M6 counterbore holes are placed and they appear in de FeatureManager Design Tree.



Chapter 7: PhotoWorks

With PhotoWorks, you can create realistic images of parts and assemblies. By editing the models characteristics, you give the model a realistic appearance. For a good result, the model needs an environment, the so-called scene. You can create your own scene, with the desired dimensions and materials, and create your own light settings to accentuate certain objects or to make things visible.

Enabling PhotoWorks

To work with Photoworks you first have to enable it.

1. Click **Tools, Add-ins**.

2. In the dialog box, select **Photoworks**, then click **OK**.

   For the rest of this chapter, it can be useful to make the Photoworks toolbar visible. You can enable the toolbar by selecting **View, Toolbars, Photoworks**.

Open a part

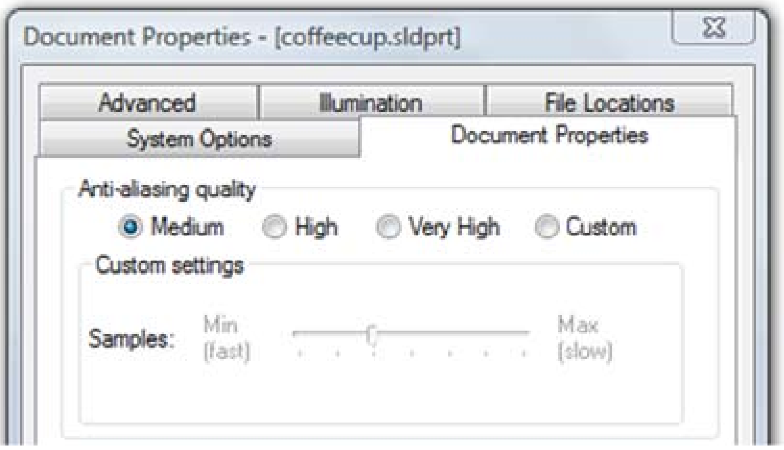
First you open the part or assembly you would like to work with, in this case it'll be a coffee cup from the tutorial folder.

3. Click **Open**, and browse to the part '**coffeecup**' in the '**Chapter 7/Coffeecup**' folder.

4. Click **Options** on the **PhotoWorks** toolbar, or click **PhotoWorks, Options**.

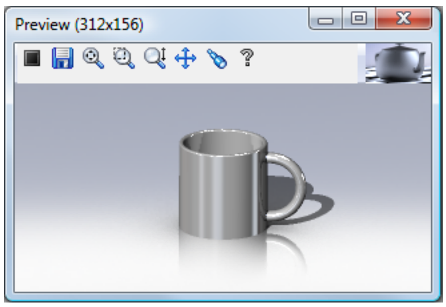
5. In the **Document Properties** tab, under **Anti-aliasing quality**, select **Medium**.
   Click **Apply**, then **Close** the dialog box.

   The anti-aliasing quality is set to medium to save rendering-time while changing the model. For creating the final render you can adjust the rendering quality to a higher setting for a better image quality.



6. Select **Photoworks**, **Preview Window.** the Preview Window appears.

   In the **Preview Window** dialog box a preview is immediatly shown of the scene. If not, click the green arrow to show the preview.

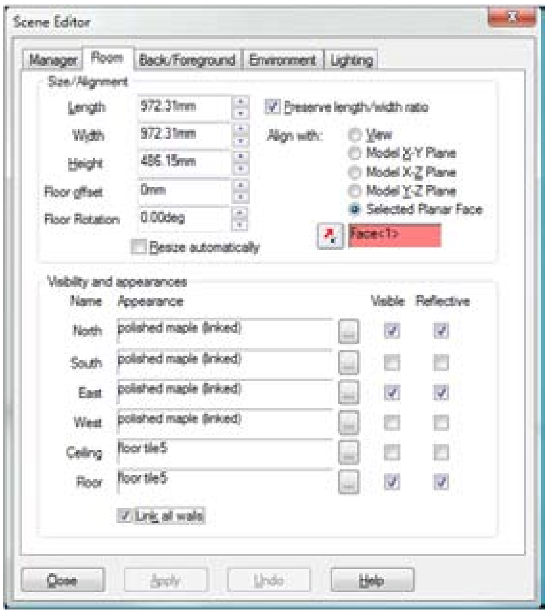


Setting a scene

If you want to create a realistic image, you need an environment around your model, a scene. This is necessary to make, for instance, reflections in the model possible. The scene consists of a virtual box or a sphere that surrounds the SolidWorks model. It's better to first create a scene, and than adjust the models properties, because the scene has great influence on the appearance of the model.

7. Click **Scene** on the **PhotoWorks** toolbar, or click **Photoworks, scene....**

8. In the **Scene Editor** dialog box, click the **Room** tab.



9. Under **Size/Alignment**:
   - Select **Preserve length/width ratio** to preserve the ratio between the floor dimensions in a cubical scene.
   - Set **Length** to **530** and press **Enter**, the **Width** value also changes.
   - Set **Height** to **360**.
   - Set **Floor offset** to **0**.
   - Deselect **Resize automatically**.
   - When you set **Floor offset** to a negative value, the floor moves up so the coffee cup rests directly on the floor.
   - Select **Selected Planar Face** under **Align with** to align the scene with the model. Then select the bottom face of the mug.

10. Under **Visibility and materials**, in the **Floor** row, click ... .

    The Material PropertyManager for the Floor appears to the left. Also, the Appearances appear on the resources tab to the right.

11. In the resources tab to the right:
    - Expand **Stone**, then expand **Architectural**, then click **Floor Tile**.
    - Click **floor tile5**.
    - In the feature manager (left side of the screen) click **Advanced**.
12. In the PropertyManager to the left:
    - go to the tab **Mapping**.
    - Select **Fit width to selection**.
    - Click **OK**.

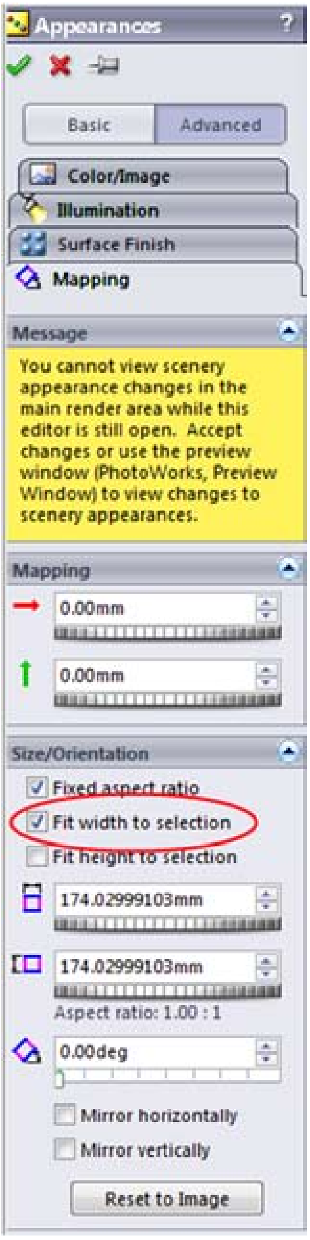
13. In the **North** row click ... .

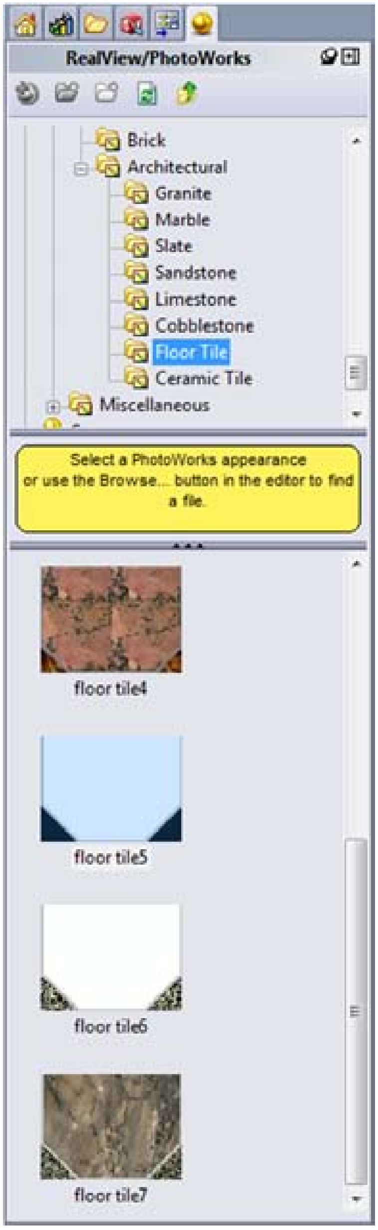
14. In the dialog box:
    - Expand **Organic**, then expand **Wood**, then click **maple**.
    - Click **polished maple**, then click **OK**.

    The material appears in the **Preview** dialog box.

15. Select **Link all walls** so the changes and materials you apply to one wall also apply to the others. The **North**, **South**, **East**, and **West** walls update to the **polished maple** material. Alse select the boxes **Visible** and **Reflective**.

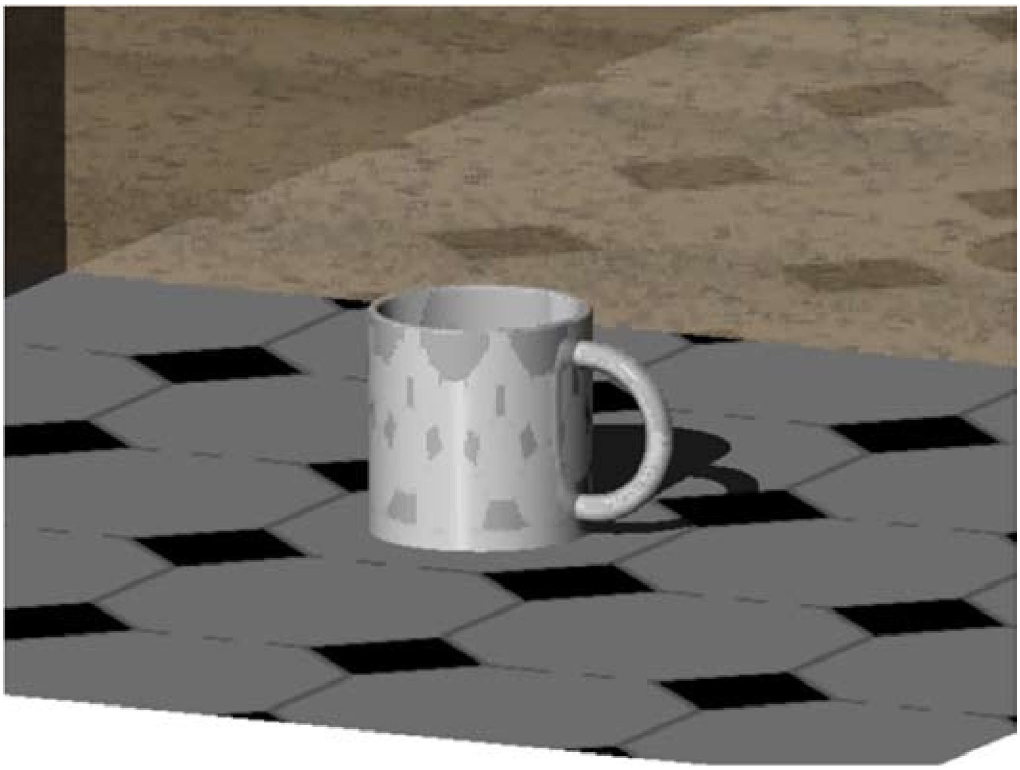
16. Click **Apply**, then click **Close**.





17. Click **Render** on the **PhotoWorks** toolbar, or click **Photoworks, Render...**

    PhotoWorks applies the scene you selected and applies the default material to the coffee cup.



Changing the lighting

The lightning is of great influence on the image, it can be used to create more depth, and is of great importance concerning visibility and reflections.

18. In the FeatureManager design tree , double-click the **Lights, cameras and scene** folder .

19. Double-click **Ambient** .

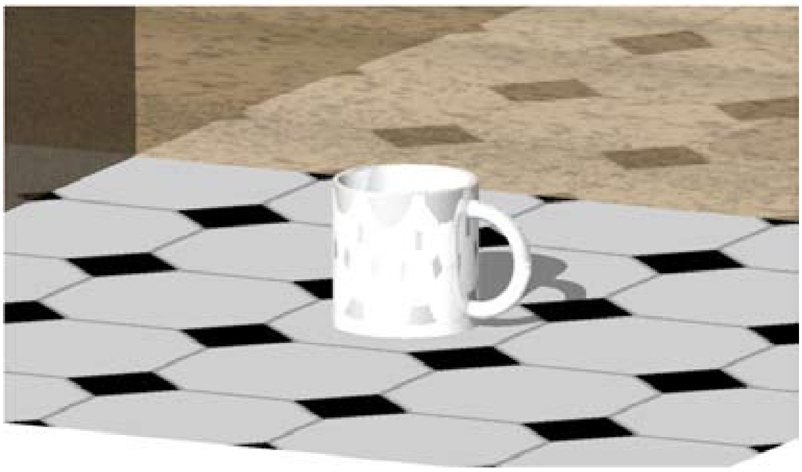
20. In the PropertyManager, type **0.5** for **Ambient** , then click **OK**.

21. Double-click **Directional1** .

22. In the PropertyManager, under **Basic**, set the **Brightness** to 0.7. Under **Light Position**, set **Longitude** to **-17** and **Latitude** to **16**, then click **OK**.

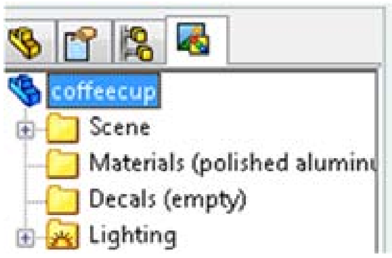
    This positions the light to appear from behind and left of the viewer.

23. Click **Render** on the **PhotoWorks** toolbar, or click **Photoworks, Render...**

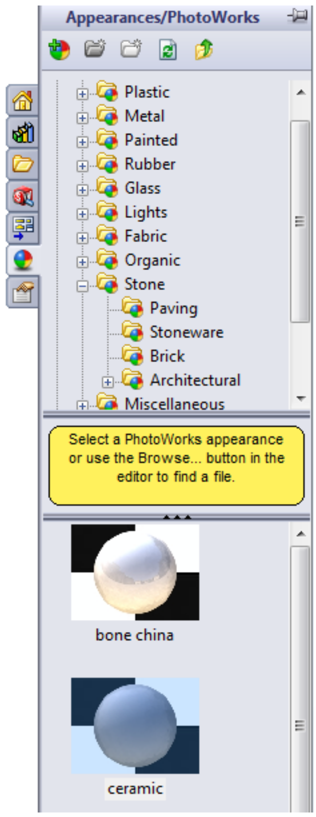


Adding the material

24. Click **coffeecup** at the top of the **Render Manager** tab . Click **Appearance** on the **(Heads-up) View** toolbar, or click **Photoworks, Appearance....**



25. In the dialog box, expand **Stone**, then expand **Stoneware**, then click **ceramic**. Click **OK**.

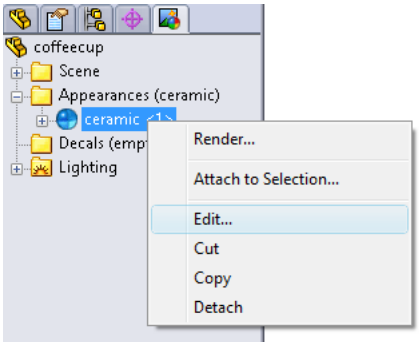


26. Click **Render** on the PhotoWorks toolbar, or click **Photoworks, Render...**
 The coffee cup is rendered in ceramic material.

Changing the color

27. Expand **Appearance** in the **Render Manager** tab .

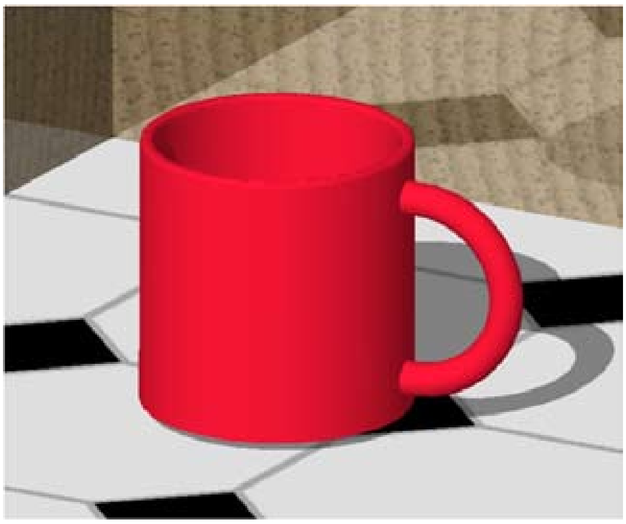
28. **Right**-click **ceramic** and select **Edit.**



29. In the color dialog box to the left:
   - Type **240** for **Red**, **20** for **Green**, and **50** for **Blue**.
   - Click **OK** to apply the new color.

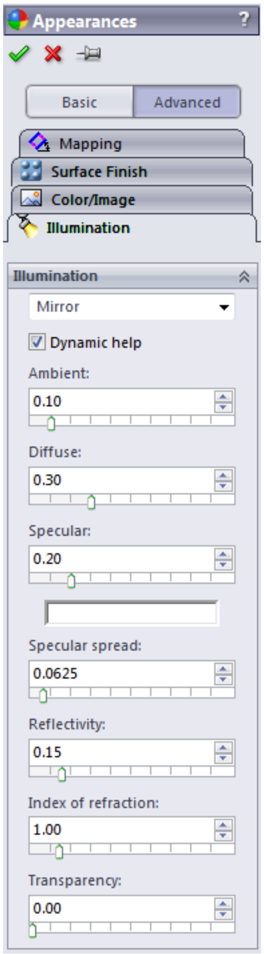
30. Click Render on the PhotoWorks toolbar, or click **Photoworks, Render...**

    The coffee cup is rendered in your custom color.



Editing the material

Together with shape and color, texture is an important characteristic of an object. To make the image more realistic you can edit the material to approximate the appearance of the applied material.



31. Double-click **Ceramic** under **Appearances** on the **Render Manager** tab .

32. On the **Illumination** tab of the **Material** Editor:
    • Select **Mirror** in **Material type** to choose a highly reflective material.
    • Type **0.3** for **Diffuse**.
    • Type **0.2** for **Specular**. Diffuse and Specular control the intensity of the light on the surface.
    • Type **0.15** for **Reflectivity** to adjust the reflectivity of the material.
    • Click **OK**.

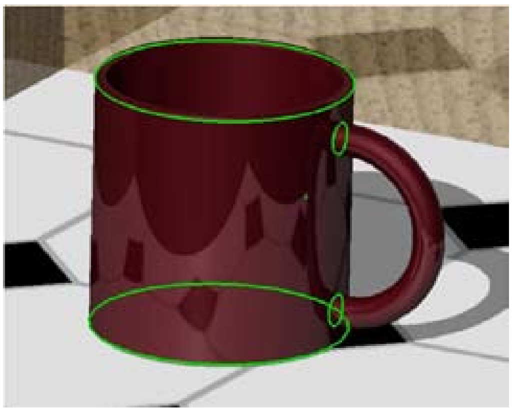
33. Click **render** on the **PhotoWorks** toolbar, or click **Photoworks, Render...**

Now you can see a realistic reproduction of, for example, the reflection of the floortiles in the cup.

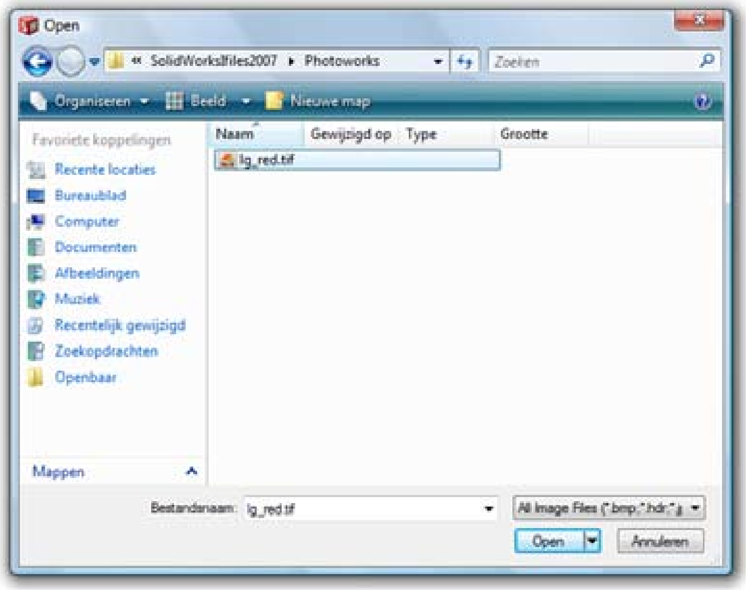
Applying a decal

Now you are going to apply a decal, as often used for advertisement purposes, on your cup. An image will be mapped onto the outer surface of the cup. This type of mapping can be used to place text on an object, but also to give an object a certain texture.

34. Select the surface you want to place the decal on, in this case the outer face of the coffeecup.



35. Click **Decal** on the **Photoworks** toolbar, or click **Photoworks, Decal...**

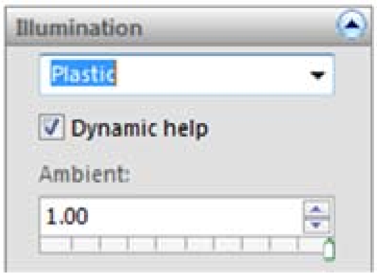


36. Click on the **Browse** button in the **Decals** property manager and browse to: **lg_red.tif** in the '**Chapter 7/Coffeecup**' folder. **Open** the image.

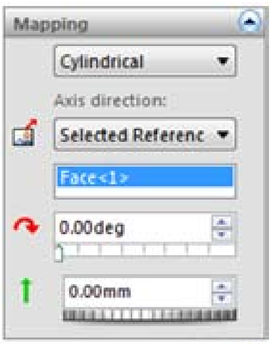
37. Click **Save Decal...** to save the decal. Select the folder where you want to save the decals and click **Save**. A message appears with the question if you want to make the folder visible for Photoworks, click **yes**.

38. Under **Mask Image**, select **Image mask file**. Click on the **Browse** button below '**Mask file path**', browse to '**lg_redmask.bmp**' and click on **Open**.

Because the decal is white, the image shown in **Resulting Image** is hard to see.

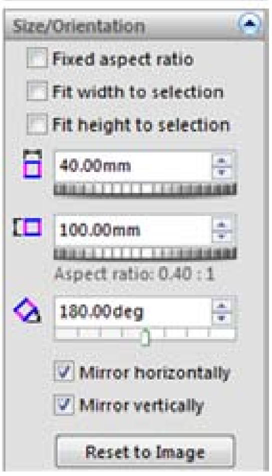


39. On the **illumination** tab:
   - Select **Plastic** in **Material type**.
   - Type **0.75** for **Ambient**.



40. On the **Mapping** tab:
   - Under **Mapping type**, select **Cylindrical**.
   - Clear **Fixed Aspect Ratio**.
   - Under **Mapping**, set **About axis** to **100 deg**, and set **Along axis** to **-0.2 mm**.
   - Under **Size/Orientation**, set **Width** to **40**, and set **Height** to **100**.

41. Click **OK**.



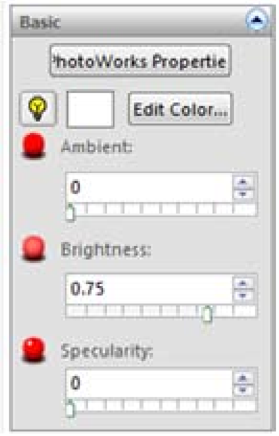
42. Click **render** on the **PhotoWorks** toolbar, or click **Photo-works, Render...**

Add a Spot light

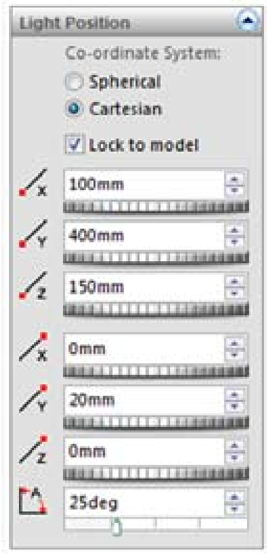
Now you are going to place a spot light on the coffee cup, to make it stand out more.

43. Right-click the **Lights, Cameras and Scene** folder in the FeatureManager design tree , then select **Add Spot Light**.

44. in the PropertyManager, under **Basic**, type **0.75** for **Brightness** .



45. Under **Light Position**:
    - Select **Lock To Model**. When you lock the light position to the model, the position of the light relative to the model is fixed. So when you move or rotate the model, the light moves along with it.
    - Set the upper (source-)coordinates :
        - **X** to **100**
        - **Y** to **400**
        - **Z** to **150**
    - Set the lower (target-)coordinates :
        - **X** to **0**
        - **Y** to **20**
        - **Z** to **0**
    - Set the **cone angle** to **25 deg**.



***Tip:** Press **Enter** to see the point light in the graphics area. The point light appears only if the model is not currently rendered.*

46. Click **OK**.

47. Click **render** on the **PhotoWorks** toolbar, or click **Photoworks, Render...**

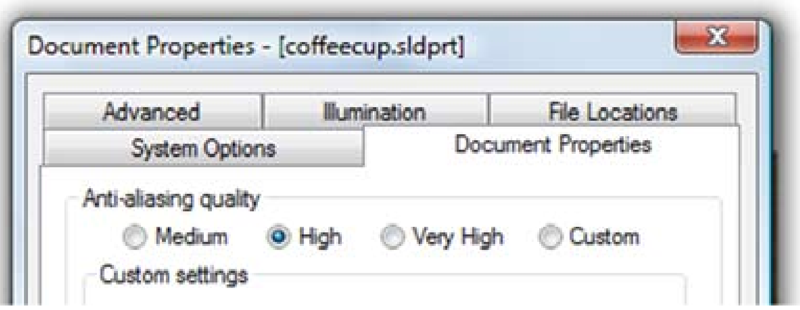
The coffee cup is now in the spotlight.

Rendering the coffee cup

Earlier you set the anti-aliasing quality to medium to save rendering time. Now we are ready editing the model and its environment, and want to render the image in full quality.

48. Click **Options** on the **PhotoWorks** toolbar, or click **PhotoWorks, Options**.

49. On the **Document Properties** tab, under **Anti-aliasing quality**, select **High**. Click **Apply**, then **Close** the dialog box.



50. Click **render** on the **PhotoWorks** toolbar, or click **Photoworks, Render...**

Now the model is rendered in high quality, with smooth edges and a finished look.

51. Exit the assembly, saving your work.