An Overview of SolidWorks and Its Associated Analysis Programs

prepared by Prof. D. Xue
University of Calgary

- SolidWorks - a solid modeling CAD tool.
- COSMOSWorks - a design analysis system fully integrated with SolidWorks, providing one screen solution for stress, frequency, buckling, thermal, and optimization analyses.
- COSMOSXpress - an easy-to-use stress analysis tool
- COSMOSMotion – a mechanical system simulation software.

SolidWorks

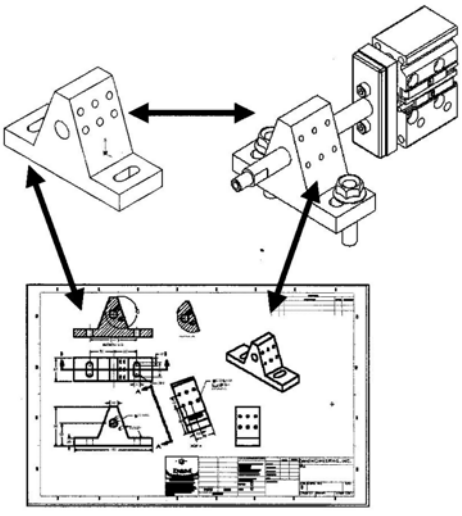
1. Introduction

(1) What is SolidWorks?

A Design Automation Software Package Used to Produce

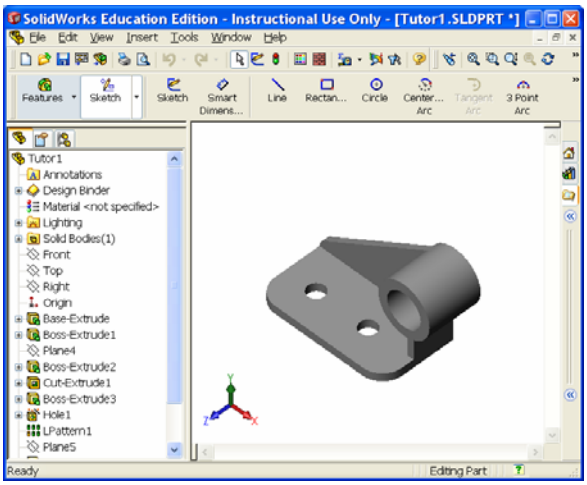
- Parts
- Assemblies
- Drawings

(From Planchard and Planchard 2003)



(2) Starting Up

Window: Start->Programs->SolidWorks 2005-> SolidWorks 2005



(3) References

• **Online Help**

Menu: Help->SolidWorks Help Topics

• **Online Tutorial**

Menu: Help->Online Tutorial

• **Book:**

D. C. Planchard and M. P. Planchard, *Engineering Design with SolidWorks 2005*, SDC Publications, 2005

D. C. Planchard and M. P. Planchard, *Engineering Design with SolidWorks 2004*, SDC Publications, 2004

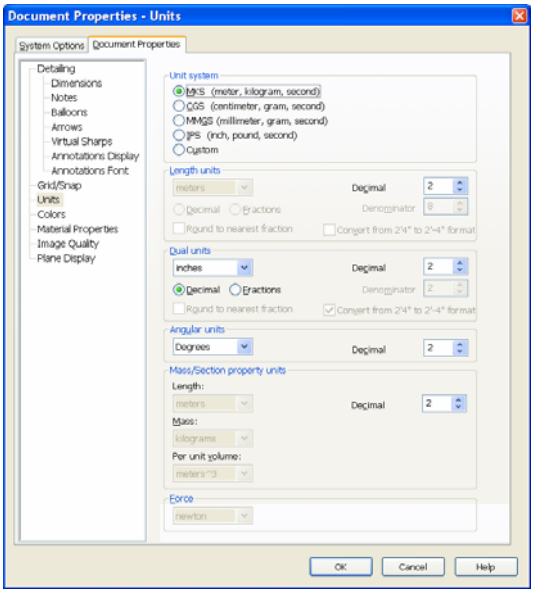
D. C. Planchard and M. P. Planchard, *Engineering Design with SolidWorks 2003*, SDC Publications, 2003

(4) SolidWorks Model Types

Type	Function	Data File
Part	3-D Object	*.SLDPRT
Assembly	Many Parts	*.SLDASM
Drawing	Multi-views	*.SLDDRW

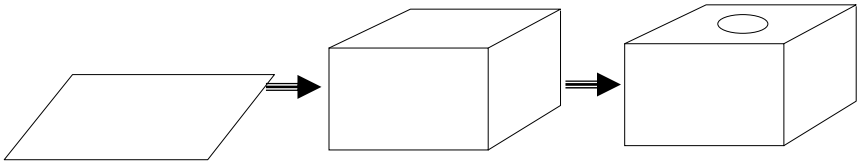
e.g.,
Base.sldprt
Base-Rod.sldasm
Base.slddrw

2. Part Modeling
(1) Setting Up Unit
Menu: Tools->Options



(2) 3-D Object Creation Procedure

By Creating Features



Top Plane + Block – Hole

Each Feature:

- 2-D Sketching
- 3-D Formation

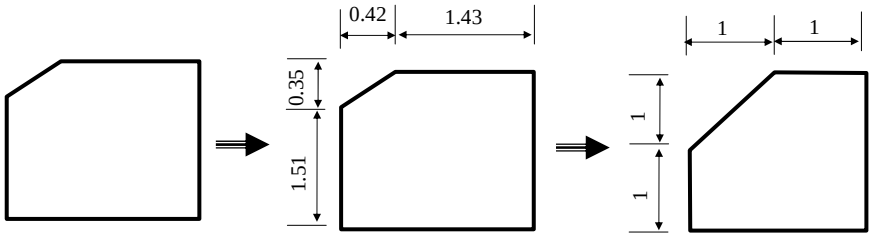
(3) 2-D Sketching

Parametric Modeling

(a) Procedure

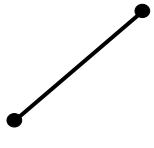
- Sketch the geometry
- Dimension the geometry
- Modify the dimension values

e.g.,

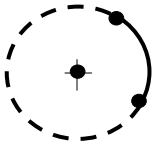


(b) 2-D Object Creation Methods

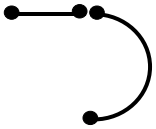
Menu: Tools->Sketch Entities



Line

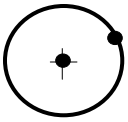


Centerpoint Arc



Tangent Arc

3 Point Arc



Circle

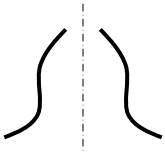
Spline

Rectangle

Point

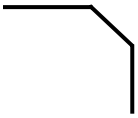
(c) Additional 2-D Object Creation Methods

Menu: Tools->Sketch Tools

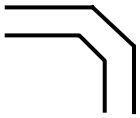


Mirror

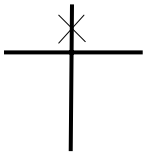
Fillet



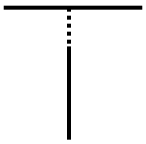
Chamfer



Offset Entities

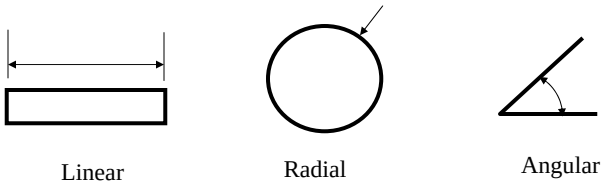


Trim



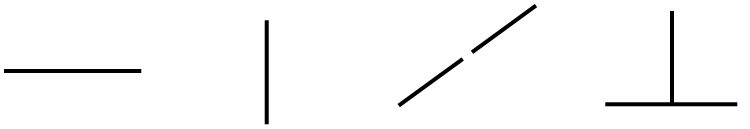
Extend

(d) Dimensioning
Menu: Tools->Dimensions

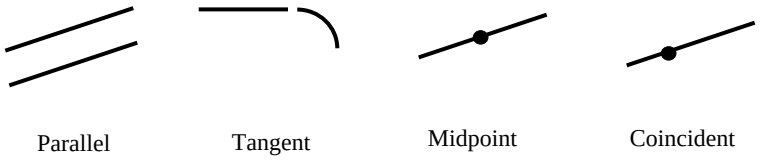


Linear Radial Angular

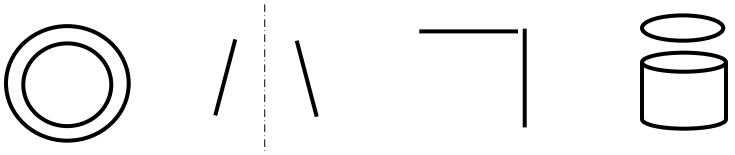
(e) Relations
Menu: Tools->Relations



Horizontal Vertical Collinear Perpendicular

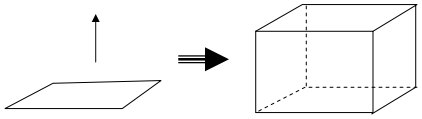


Parallel Tangent Midpoint Coincident

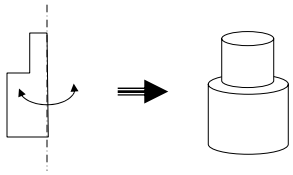


Concentric Symmetric Equal Coradial

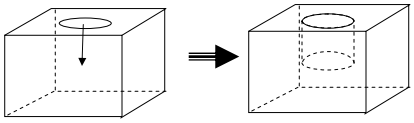
(4) Features



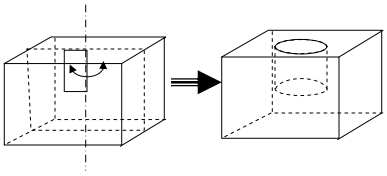
Extruded Boss/Base



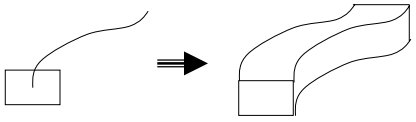
Revolved Boss/Base



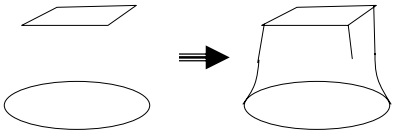
Extruded Cut



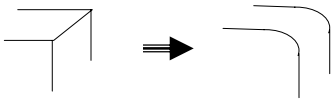
Revolved Cut



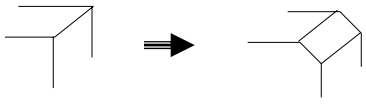
Sweep Boss/Base



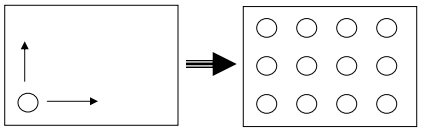
Lofted Boss/Base



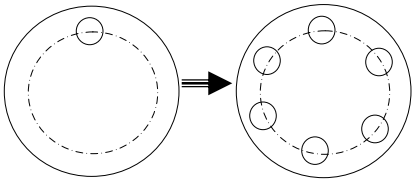
Fillet



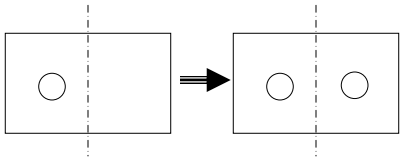
Chamfer



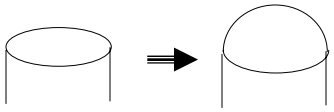
Linear Pattern



Circular Pattern

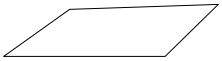


Mirror

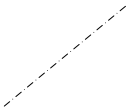


Dome

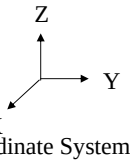
(5) Reference Geometry



Plane

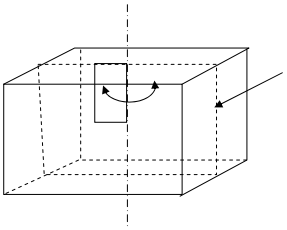


Axes



Coordinate System

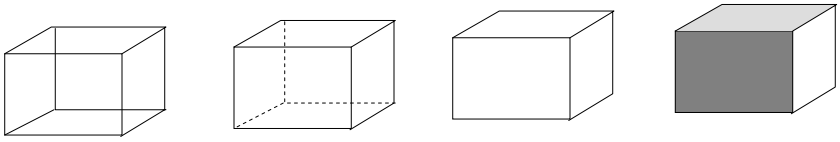
e.g.,



A reference plane for creating a sketch of revolved cut feature

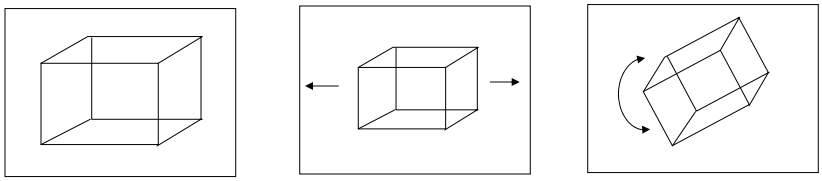
(6) Viewing

Menu: View->Display



Wireframe | Hidden Lines Visible | Hidden Lines Removed | Shaded With Edges

Menu: View->Modify

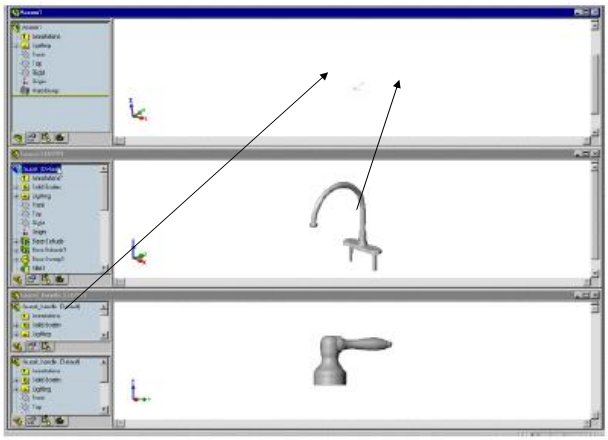


Zoom to Fit | Pan | Rotate

3. Assembly Modeling

(1) Loading the Components

Menu: Insert->Component->Existing Part/Assembly

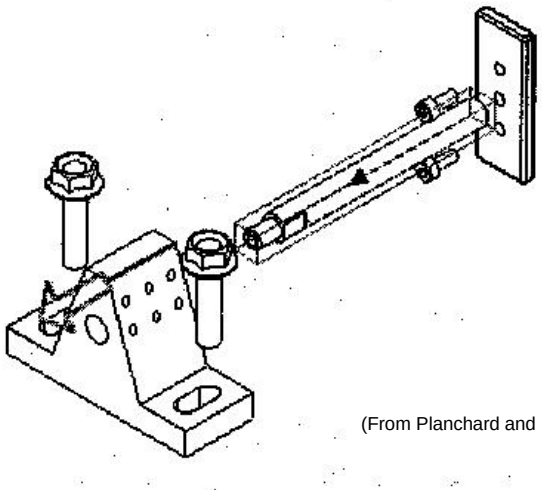


(2) Defining Mates

Menu: Insert->Mate

(3) Exploded View

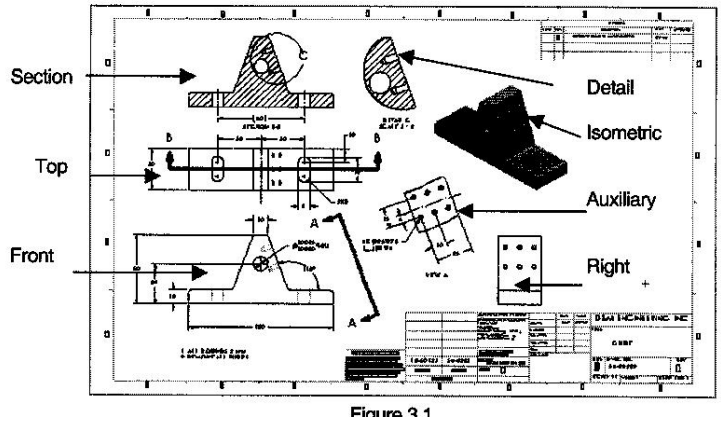
Menu: Insert->Exploded View



(From Planchard and Planchard 2003)

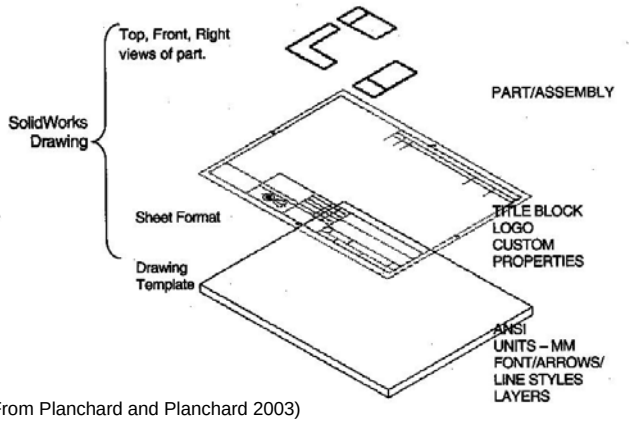
4. Drawing Modeling

2-D Drawing of a Part or an Assembly



(From Planchard and Planchard 2003)

(1) Drawing Template and Drawing Format



(From Planchard and Planchard 2003)

Orientation
- Landscape
- Portrait

Size
- A
- B
- A4
- A3

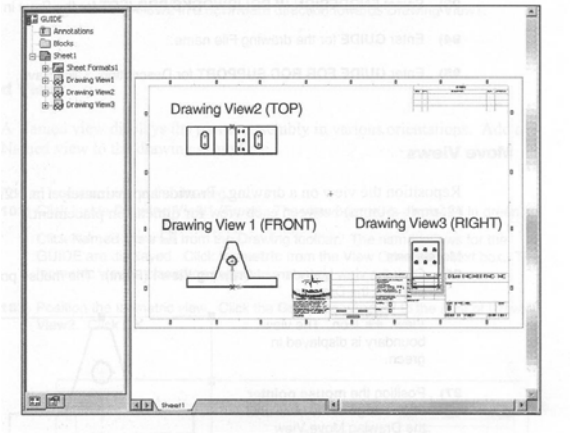
... ...

Menu: File->New->Draw

(2) Creating Views

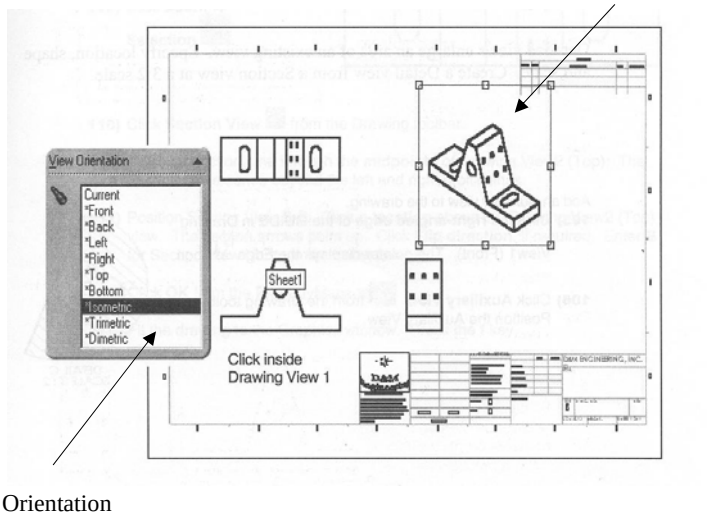
Menu: Insert->Drawing View

• Standard 3 View



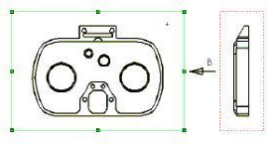
(From Planchard and Planchard 2003)

• Model View

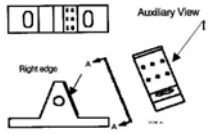


(From Planchard and Planchard 2003)

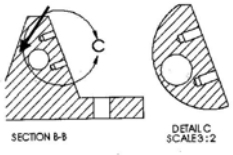
• Derived Drawing Views



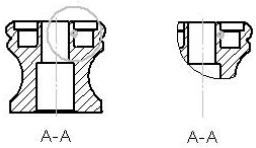
Projected View



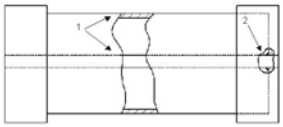
Auxiliary View



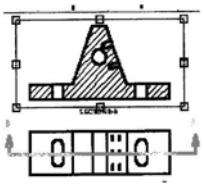
Detail View



Crop View



Broken-Out Section



Section View

(3) Dimensions

Menu: Tools->Options

Select Styles of Font, Leader, Precision, Tolerance, Arrow, etc.

Two Ways to Create Dimensions

(i) Display All Dimensions and Then Modify These Dimensions

Menu: Insert->Model Items

Select Checkboxes

• Dimension

• Import items into all views

(ii) Create Required Dimensions Manually

Menu: Tools->Dimensions

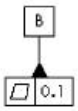
(4) Annotations

Menu: Insert->Annotations

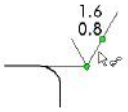
3 x 25 ABC

Note

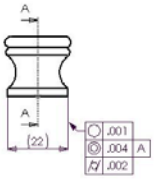
Balloon



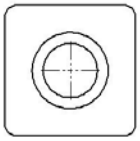
Datum Feature Symbol



Surface Finish Symbol



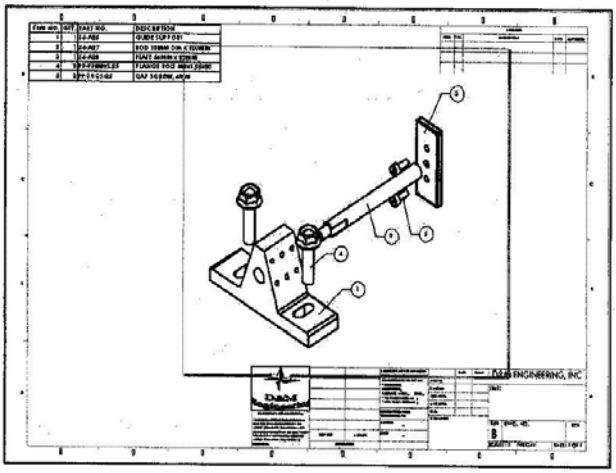
Geometric Tolerance



Center Mark

(5) Bill of Materials

Menu: Insert->Tables->Bill of Materials

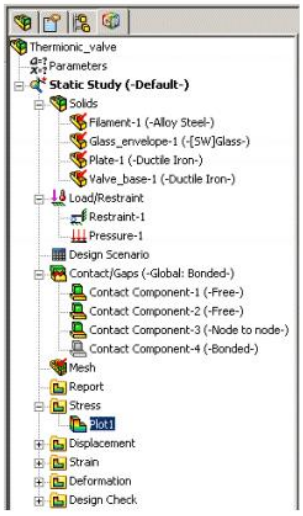


(From Planchard and Planchard 2003)

COSMOSWorks

COSMOSWorks is a design analysis system fully integrated with SolidWorks. COSMOSWorks provides one screen solution for stress, frequency, buckling, thermal, and optimization analyses.

COSMOSWorks Manager Tree



Structure Analysis Steps

1. Create a study defining its analysis type and options.
2. If needed, define parameters of your study.
3. Define material properties.
4. Specify restraints.
5. Specify the loads.
6. Mesh the model.
7. Link the parameters to the appropriate study inputs.
8. If desired, define up to 100 design scenarios.
9. Run the study or selected design scenarios.
10. View and list the results.

Design Studies

(1) Analysis Types

- **Static**: Linear static analysis
- **Frequency**: Frequency analysis
- **Buckling**: Linearized buckling analysis
- **Thermal**: Thermal analysis
- **Optimization**: Optimization analysis

(2) Mesh Types

- Solid
- Shell mesh using mid-surfaces
- Shell mesh using surfaces

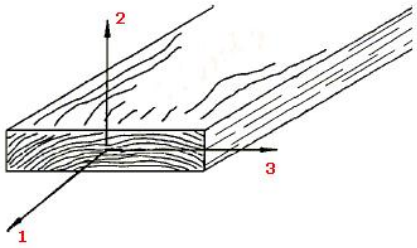
Isotropic and Orthotropic Materials

(1) Isotropic Materials

A material is isotropic if its mechanical and thermal properties are the same in all directions.

(2) Orthotropic Materials

A material is orthotropic if its mechanical or thermal properties are unique and independent in three mutually perpendicular directions.



Material Properties

- Elastic Modulus
- Shear Modulus
- Poisson's Ratio
- Coefficient of Thermal Expansion
- Thermal Conductivity
- Density
- Specific Heat

Summary of Restraint Types

Restraint Type	Load Entities
Fixed (fixes translations and rotations)	Vertices , Edges , and Faces
Immovable (fixes translations only)	Vertices , Edges , and Faces
Use Reference Plane or Axis	Vertices , Edges , and Faces
On Flat Face	Planner Faces
On Cylindrical Face	Cylindrical Faces
On Spherical Face	Spherical Faces

Structural Loads

- **Pressure** (**uniform** or **nonuniform** distribution)
- **Force** (**uniform** or **nonuniform** distribution)
- **Gravity**
- **Centrifugal Load**
- **Remote Loads** (**direct load transfer , rigid connection , remote displacement**)
- **Bearing Loads**
- **Connectors** (Rigid, Spring, Pin, Elastic Support)
- **Temperature** (prescribed temperatures, uniform temperature change, or a temperature profile from a thermal study)
- **Motion Loads from COSMOSMotion** (available from COSMOSWorks, Import Motion Loads)
- **Shrink Fitting** (applied as a **contact condition**)

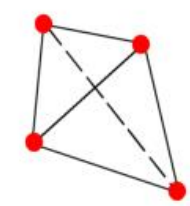
Solid Mesh

• **Draft Quality Mesh:**

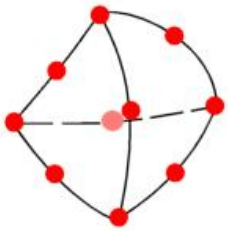
The automatic mesher generates linear tetrahedral solid elements.

• **High Quality Mesh:**

The automatic mesher generates parabolic tetrahedral solid elements.



Linear solid element



Parabolic solid element

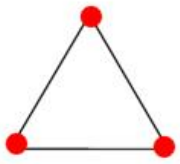
Shell Mesh

• **Draft Quality Mesh:**

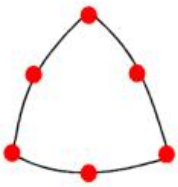
The automatic mesher generates linear triangular shell elements.

• **High Quality Mesh:**

The automatic mesher generates parabolic triangular shell elements.



Linear triangular element



Parabolic triangular element

Plotting Results

- Stress results
- Principal stresses
- Contact Pressure
- Displacement results
- Strain results
- Deformation results
- Thermal results
- Fatigue results

Listing Results

- Listing Stress
- Listing Contact/Friction Forces
- Listing Pin/Bolt Forces
- Listing Displacement
- Listing Reaction Forces
- Listing Interface Forces
- Listing Strain
- Listing Modes
- Listing Mass Participation Ratios
- Listing Thermal Results
- Listing Results on Selected Entities of the Model

References

• **Online Tutorial:**

Menu: Help->COSMOSWorks Online Tutorial

• **Online Help:**

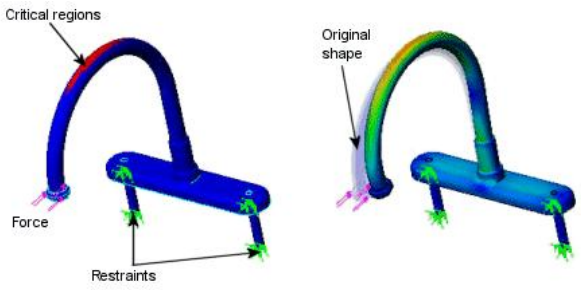
Select Windows Menu:

Programs -> COSMOS 2005 Applications -> COSMOSWorks Documentation -> English -> Online Help

COSMOSXpress

An Easy-to-use Stress Analysis Tool

An Example:



Critical regions Deformed shape

Stress Analysis

Stress or static analysis calculates the displacements, strains, and stresses in a part based on material, restraints, and loads.

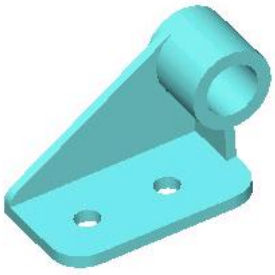
Why Analyze?

After building your design in SolidWorks, you may need to answer questions like:

- Will the part break?
- How will it deform?
- Can I use less material without affecting performance?

Finite Element Method (FEM)

- FEM replaces a complex problem with many simple problems.
- It divides the model into many small pieces of simple shapes called elements.



CAD model of a bracket

Model subdivided into small pieces (elements)

Assumptions of Linear Static Analysis

(1) Linearity Assumption

The induced response is directly proportional to the applied loads. For example, if you double the magnitude of loads, the model's response (displacements, strains, and stresses) will double.

(2) Elasticity Assumption

The part returns to its original shape if the loads are removed (no permanent deformation).

(3) Static Assumption

Loads are applied slowly and gradually until they reach their full magnitudes.

Analysis Steps

(1) Define material of the part

(2) Apply restraints

(3) Apply loads

(4) Analyze the part

(5) View the results

Assigning Material

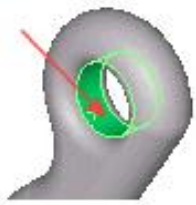
(1) To assign material from the material library

(2) To define material properties:

- **EX** (Elastic Modulus).
- **NUXY** (Poisson's ratio). If you do not define NUXY, COSMOSXpress assumes a value of 0.
- **SIGYLD** (Yield Strength). Used to calculate **factor of safety** .

Applying Restraints

- Each restraint can contain multiple faces.
- The restrained faces are constrained in all directions.
- You must at least restrain one face of the part to avoid analysis failure due to rigid body motion.

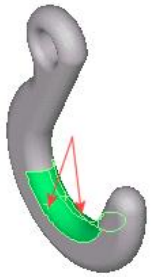


Applying Loads

(1) Forces

You can apply multiple forces to a single face or to multiple faces.

- Normal to each selected face
- Normal to a reference plane



(2) Pressure

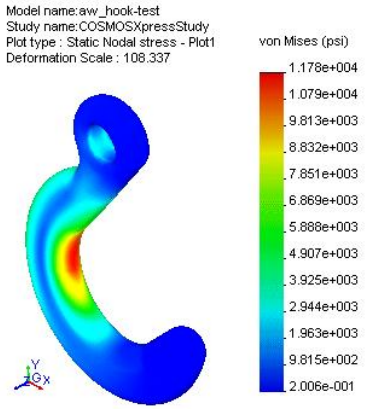
You can apply multiple pressures to a single face or to multiple faces. COSMOSXpress applies pressure loads normal to each face.

Analyzing the Part

- **Yes (recommended)** to accept the default mesh settings (default element size and tolerance values)
- **No, I want to change the settings** to change the default mesh settings.
  - Element Size
  - Element Tolerance

Viewing the Results

- The Stress Distribution in the Model
- The Deformed Shape of the Model



What More Can I Do With COSMOSWorks?

- **Analysis of assemblies:**

In addition to analyzing parts, you can analyze assemblies. You can assign a different material for each component.

- **Stress analysis with contact conditions:**

Friction and large displacement options are supported by contact analysis.

- **Shell modeling of sheet metal and thin parts:**

COSMOSWorks uses small number of shell elements instead of a large number of tetrahedral elements to mesh thin parts .

Analysis Types

In addition to extensive stress analysis options in every step of the design analysis process, COSMOSWorks offers the following additional types of analyses:

- Frequency (modal) analysis
- Buckling analysis
- Thermal analysis
- Optimization analysis
- Other types of analyses

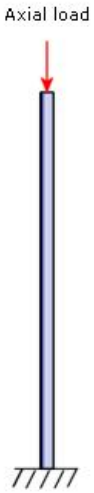
Frequency Analysis

- A body disturbed from its rest position tends to vibrate at certain frequencies called natural, or resonant frequencies. For each natural frequency, the body takes a certain shape called mode shape. Frequency analysis calculates the natural frequencies and the associated mode shapes.
- Frequency analysis can help you avoid resonance by calculating the resonant frequencies. It also provides information to solve **dynamic response** problems.

Buckling Analysis

Slender models subject to compressive axial loads tend to undergo sudden large lateral deformation at certain load levels. This phenomenon is called buckling. In some cases, buckling occurs before the material fails due to high stresses.

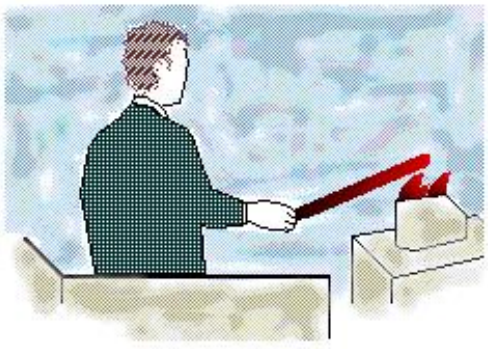
COSMOSWorks helps you avoid buckling failure by calculating minimum loads that cause buckling.



Thermal Analysis

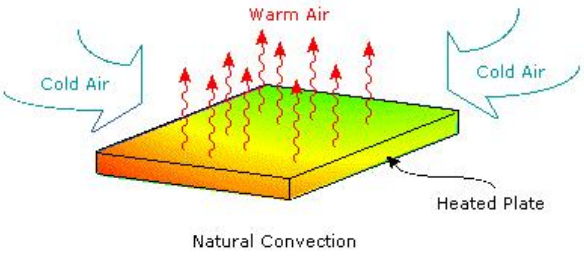
(1) Conduction:

Conduction is the transfer of heat by means of molecular agitation within a material without any motion of the material as a whole.



(2) Convection:

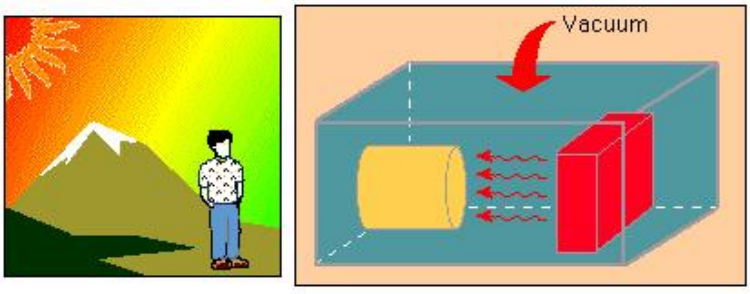
Convection is the transfer of heat by means of moving fluids.



Natural Convection

(3) Radiation:

Radiation is the transfer of heat by means of electromagnetic waves.



Optimization Analysis

• **Objective:**

State your objective. For example, minimum material.

• **Design Variables or Geometry Constraints:**

Select the dimensions that can change and set their ranges. For example, the diameter of a hole can vary from 0.5" to 1.0" while the extrusion of a sketch can vary from 2.0" to 3.0".

• **Behavior Constraints:**

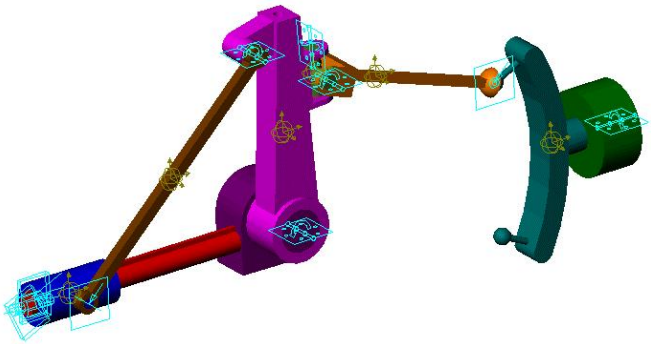
Set the conditions that the optimum design must satisfy. For example, stresses, displacements, temperatures should not exceed certain values and the natural frequency should be in a specified range.

Other Types of Analysis

- Nonlinear static and dynamic stress analysis
- Dynamic response analysis
- Fluid flow analysis (COSMOSFloWorks)
- Motion simulation (COSMOSMotion)
- Electromagnetic analysis (COSMOSEMS)

COSMOSMotion

COSMOSMotion is design software for mechanical system simulation.



IntelliMotion Browser

Primary Type Node	Secondary Type Node
Assembly Components	No secondary types
Parts	Moving Parts
	Ground Parts
Constraints	Joints
	Contact
	Couplers
Forces	Springs
	Dampers
	Action Only
	Action/Reaction
Results	Cplr Curve
	Linear Disp
	Angular Disp
	Velocity
	Acceleration
	Reaction Forces (Motion Only)
	XY Plots

Motion Analysis Steps

(1) Review your product concept

(2) Add constraints to define assembly movement

(3) Apply motion to the constraints in your mechanism

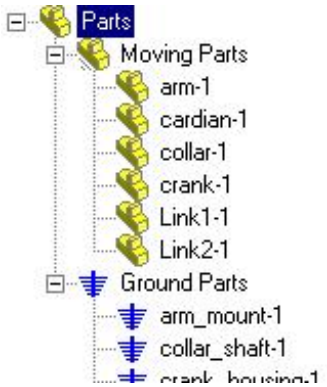
(4) Add applied loads (optional, COSMOSMotion only)

(5) Run a simulation of the mechanism

(6) Review the simulation results

Parts in Motion Analysis

- **Moving Parts:** Parts with Motion
- **Ground Parts**: Parts without Motion



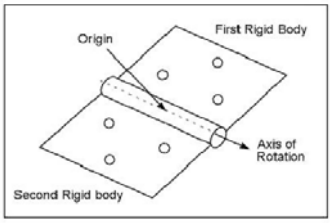
Types of Constraints

- **Joints**
- **Joint Primitives**
- **Motion Drivers**
- **Contact**

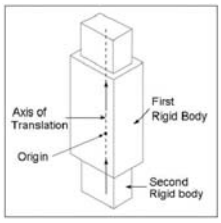
Joints and Degree-Of-Freedom (DOF)

Joint	Translational DOF	Rotational DOF	Total DOF Constrained
Revolute	3	2	5
Translational	2	3	5
Cylindrical	2	2	4
Spherical	3	0	3
Universal	3	1	4
Screw	.5	.5	1
Planar	1	2	3
Fixed	3	3	6

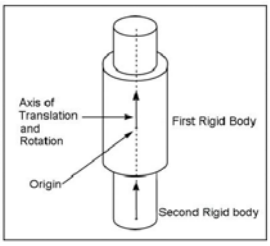
Types of Joints



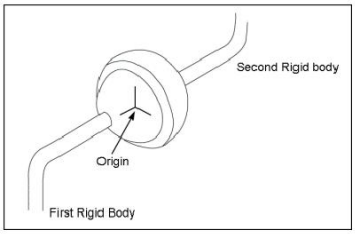
(1) Revolute Joint



(2) Translational Joint

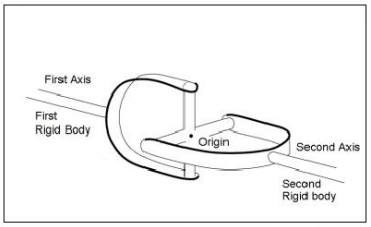


(3) Cylindrical Joint

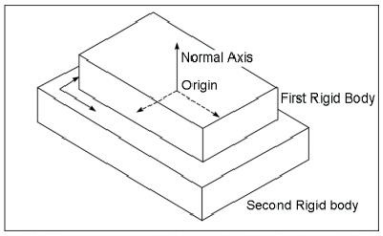


(4) Spherical Join

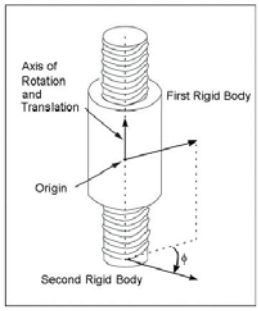
Types of Joints



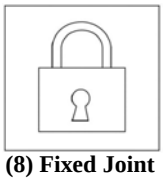
(5) Universal Joint



(7) Planar Joint



(6) Screw Joint

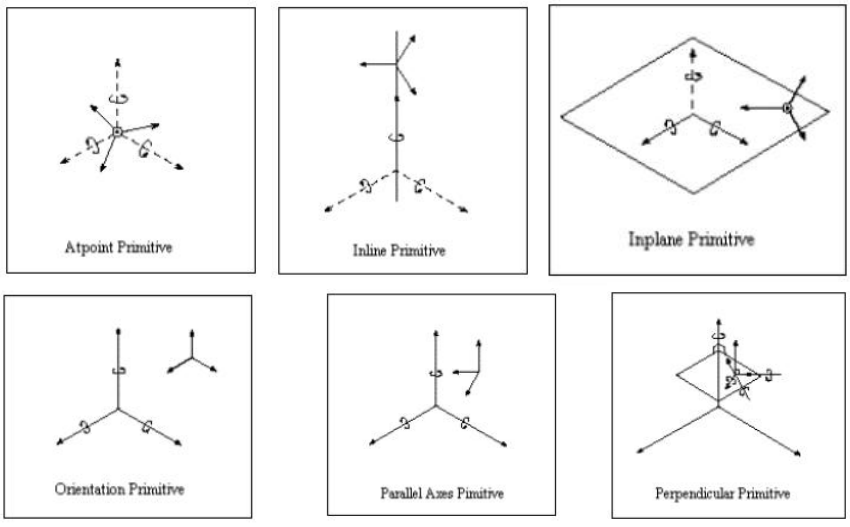


(8) Fixed Joint

Joint Primitives and DOF

Joint Primitive	Translational DOF	Rotational DOF	Total DOF Constrained
Atpoint	3	0	3
Inline	2	0	2
Inplane	1	0	1
Orientation	0	3	3
Parallel Axis	0	2	2
Perpendicular	0	1	1

Types of Joint Primitives



Motion Drivers

You add motion drivers to joints to define the movement of the joint over time.

Motion Types:

- Displacement: D(t)
- Velocity: V(t)
- Acceleration: A(t)

Contact Constraints

- **Point-curve:**

Restricts a point on one rigid body to lie on a curve on a second rigid body

- **Curve-curve:**

Constrains one curve to remain in contact with a second curve

Constraint	Translational DOF	Rotational DOF	Total DOF Constrained
Point-curve	2	0	2
Curve-curve	2	0	2

Types of Forces

- **Applied Forces:**
  - Action-only force
  - Action-only moment
  - Action/reaction force
  - Action/reaction moment
  - Impact force
  - **Flexible Connectors:**
  - Translational Springs
  - Torsion Springs
  - Translational Dampers
  - Torsion Dampers
  - Bushings
- **Gravity**

Simulation Settings

- Number of Frames
- Duration or Time Increment
- Animate during Simulation
- Use Mass Properties Stored with Parts

Solver Parameters:

Integrator Type, Maximum Iterations, Initial Time Step, Maximum Time Step, Minimum Time Step, Accuracy, Jacobian Pattern, Adaptivity

Reviewing Results

- Play animations
- Check for interference as the parts move
- Display result symbols on screen
- Plot numerical data
- Use part force results in COSMOSWorks
- Export your animations to AVI movies and VRML format
- Export force results to Excel, text file, or other FEA applications

References

- **Online Tutorial:**

Select Windows Menu:

Programs -> COSMOS 2005 Applications -> COSMOSMotion Documentation -> English -> Online Tutorials

- **Online Help:**

Select Windows Menu:

Programs -> COSMOS 2005 Applications -> COSMOSWorks Documentation -> English -> Online Help