CREATING A FINITE ELEMENT MODEL WITH THE MSC/ARIES APPLICATION MODULES

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ABSTRACT

Software generated solid models provide complete and unambiguous visual and mathematical definition of design objects.The MSC/ARIES Solids software application module affords an easy to use, highly intuitive way of creating solid models to capture the design intent of engineers and analysts. The objectives of this presentation are to:

- introduce the techniques applied to build and modify solid models.
- describe the transition from the geometric model to the Finite Element model.
- display the results of a linear static finite element analysis.

Introduction

Solid modeling is a powerful tool that enables mechanical engineers to create physically recognizable, complete, and unambiguous computer designs of parts and assemblies.

The purpose of this paper is to acquaint the reader with the functionalities afforded by the various MSC/ARIES software modules to build and modify solid models and to prepare the models for finite element analysis.

There are four major sections:

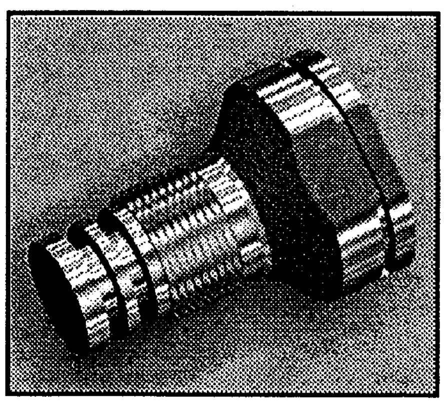
1. Discussion of the advantages of solid modeling.
2. Introduction to the basic concepts of solid modeling.
3. Step-by step description of the construction and modification of a complex solid model.
4. Transitioning the solid geometric model into a finite element model.

Solid Modeling

Solid geometry is a fairly recent addition to the repertory of computer generated graphics. Solid modeling is a very powerful tool that offers many advantages to design engineers. Some of these advantages are listed below:

- Visual realism

  Rendered images of a preliminary design communicate intent by clarifying shape, size, material, and color definitions.



- Completeness

  Solid models are topologically complete, with all exterior and interior faces present. There will be no missing faces nor gaps between faces.

- Unambiguity

  A solid model is a mathematically fully defined three-dimensional enclosed shape. Its curved surfaces, convex or concave faces, etc. are realistically presented. Physically impossible geometry, e.g. interior and exterior faces crossing, will not be created.

- Integration with other applications

  The model database of a solid model is available for use in tightly integrated analysis applications, (Mass Properties, Finite Element Analysis, Mechanisms Analysis, etc.) and for non-geometric applications, e.g. part control, administrative data.

Other Geometric Entities

In addition to solid geometry, the MSC/ARIES software also utilizes *wireframe* and *surface* geometry.

The primary function of **wireframe** entities (curves, circles, splines, lines) is to serve as the initial profiles of certain solid shapes. **Surface geometry** can be useful for modeling sheet metal objects and for generating *surface meshes* for two-dimensional analysis.

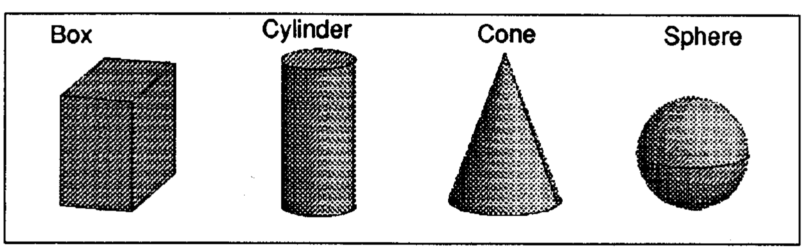
Solid Modeling Concepts

Complex solid models evolve from simple components called *primitives. Construction operations* combine, cut, or otherwise shape primitives into more complicated forms. Solid *features* can be added to enhance the design.

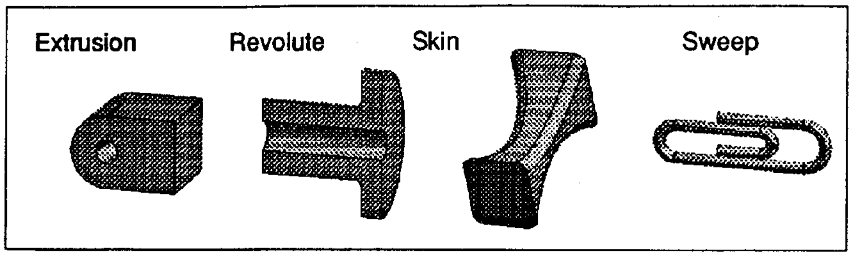
Solid Primitives

Solid primitives are simple solid objects created in one single operation. Primitives can be categorized as *shape-based* or *curve-based.*

Shape-based primitives are *box, cylinder, cone,* and *sphere.* These are predefined, unique shapes that do not require any previously created geometry. Their parameters consist of coordinate system locations and dimensions.



Curve-based primitives are *extrusion*, *revolute, skin*, and *sweep*. Their shape is defined by profile curve(s) and various operation types. Each primitive type has its own specific parameter requirements.



- **Extrusion** - a prismatic solid whose end faces are determined by one or more non-intersecting generating curves and whose connecting faces result from the curves being swept along a straight line from one end plane to the other. Optionally, extrusions may be created with a specified *draft angle*. Required input data are the generating curve(s), extrusion direction, and depth.

- **Revolute** - a solid of revolution that is the result of a generating curve being swept angularly around a selected center line. Input parameters include the definition of the generating curve and the centerline, and the angular extent of the revolution.

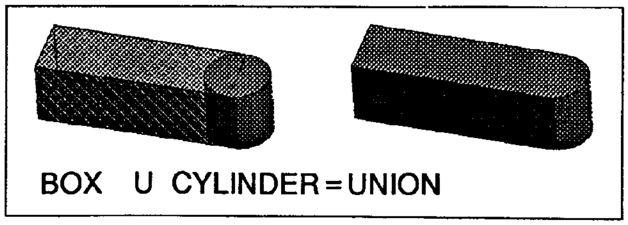
- **Skin** - a free-form, sculpted solid that is "lofted" over any number of non-coplanar curves. Required input: cross sectional curves, connectivity information, degree of path interpolation.

- **Sweep** - a solid generated by sweeping a single cross-section curve along an arbitrary path curve. User input includes definition of the cross-section curve and the path curve.

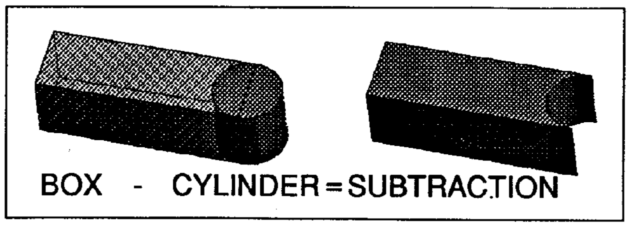
Construction Operations to Develop Complex Solids

Construction operations are high-level processes that combine primitives to create a resultant object. These operations are *union, subtraction,* and *intersection.*

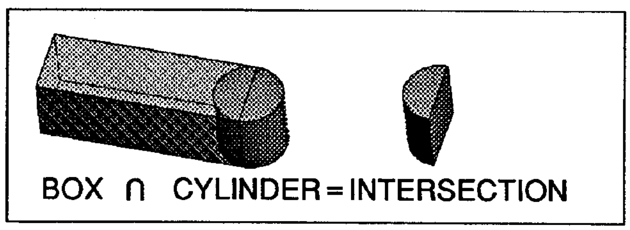
- **Union** - The resultant solid occupies the sum of all spaces occupied by the components. Space(s) common to *more than one* component are counted only once.



- **Subtraction** - The resultant solid occupies the space of the first solid less the space(s) occupied by subsequently selected solid(s).

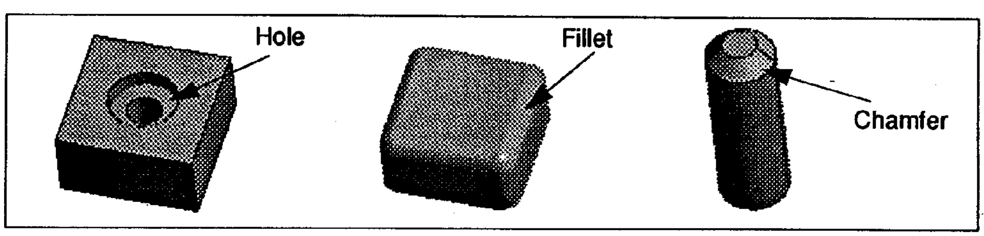


- **Intersection** - The resultant solid occupies only the space that is *common to* all component solids.



Solid Features

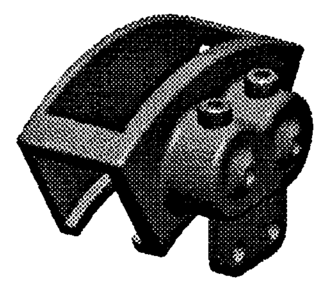
Solid features are *hole*, *fillet*, and *chamfer*. These are easily constructed through the user interface.



Building a Part from Primitives

The following example, a brake caliper shown below, illustrates the necessary steps that will create a complex part using the MSC/ARIES Solids software module.

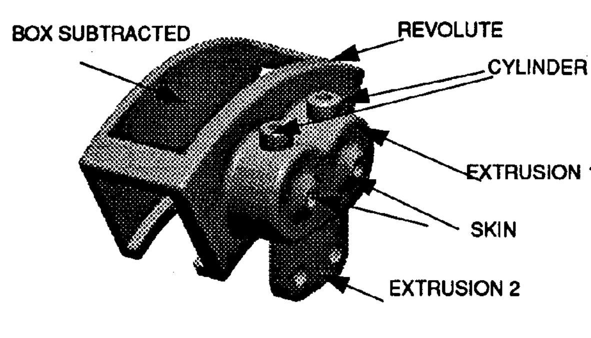
Brake Caliper



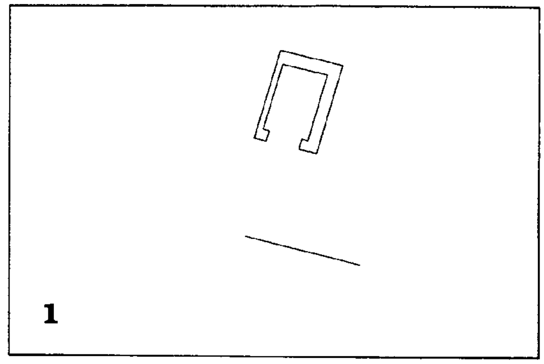
Experience with solid modeling is helpful when one is deciding what primitives and operations will be used to create a solid model. Several approaches may be explored and much depends on individual preferences. Knowledgable designers develop ways of building better, "cleaner " solid models. For example, it is advantageous to keep the number of solid components and the number of operations as low as possible.

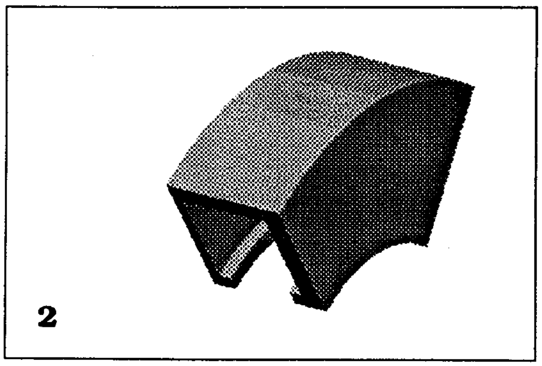
It was decided to build this part as follows:

1. Start with a solid of revolution
2. Cut a box-shape out of it
3. Union two extrusions, two cylinders, and two skinned solids to the base solid.
4. Construct fillets, chamfers, and holes.



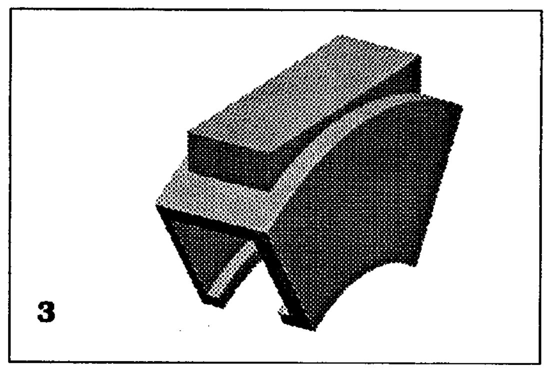
1. Start with a profile curve and center line for the REVOLUTE.

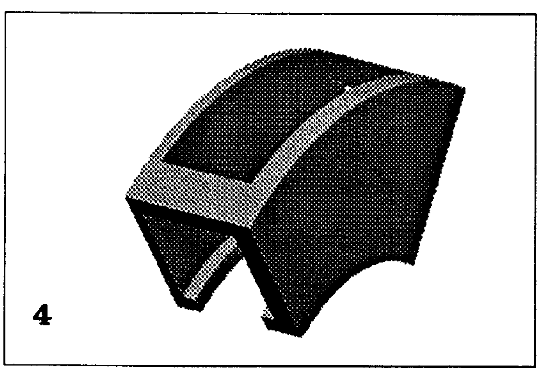




2. Add a solid REVOLUTE.

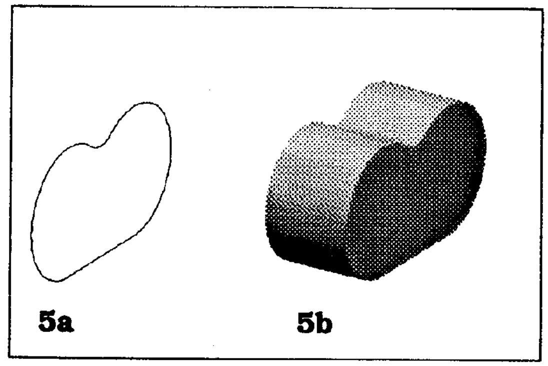
3. Add a solid BOX (this is the cutting tool).

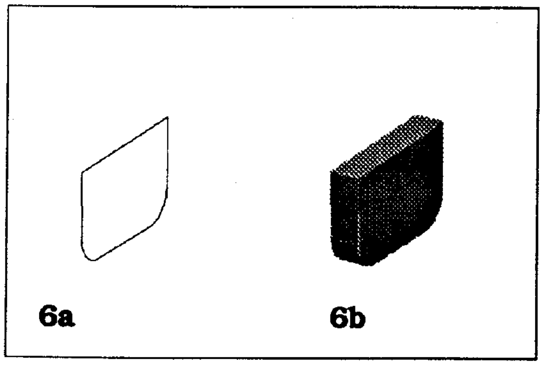




4. SUBTRACT the box from the revolute.

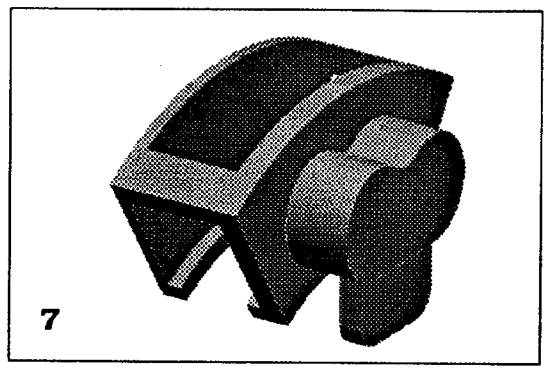
5. Make a profile curve for the first EXTRUSION (a).
Add a solid EXTRUSION (b) (detail zoomed up).

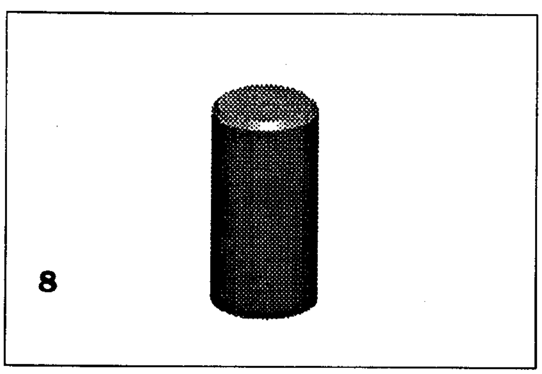




6. Make the second profile curve (a) and develop the second extrusion (b).

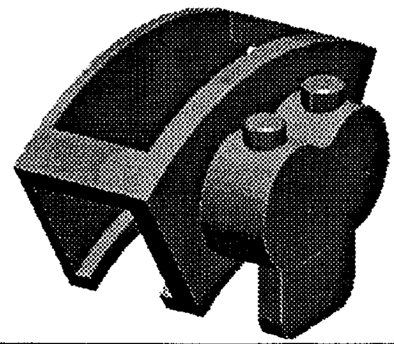
7. UNION all solids.





8. Make a CYLINDER. CHAMFER its top edges (detail zoomed up, other solid blanked).

9. MOVE and copy this component.



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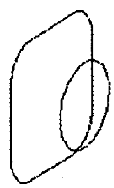

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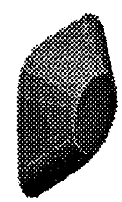
10. Create two CURVES to define a skinned solid (detail zoomed up, other solid blanked).

11. Loft a SKIN solid across the curves.



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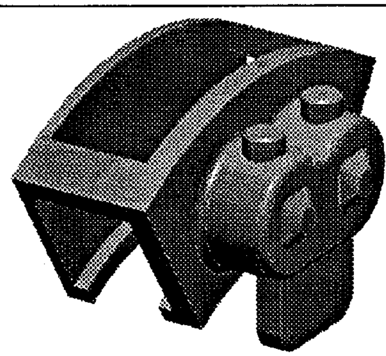

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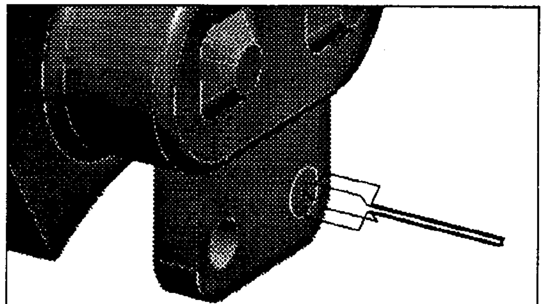
12. Move and copy the skinned solid. UNION all new components to the main body. Create all necessary FILLETS.

13. Construct the first two holes.



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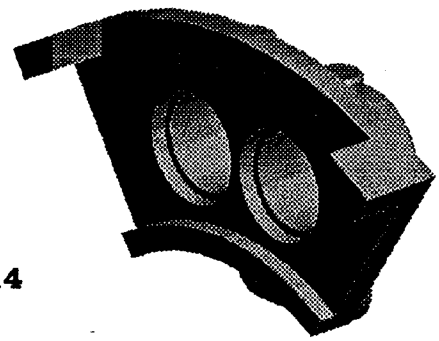

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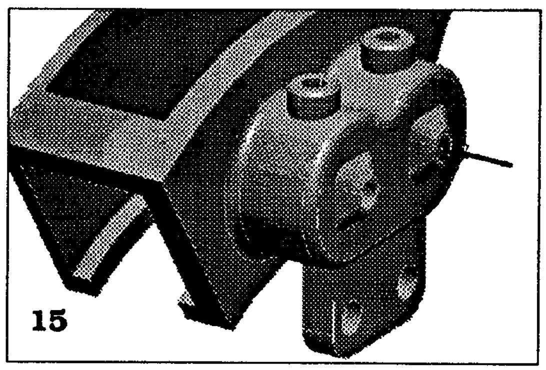
14. ROTATE the VIEW, SECTION the model, and make the holes for the pistons.

15. Make all remaining holes.



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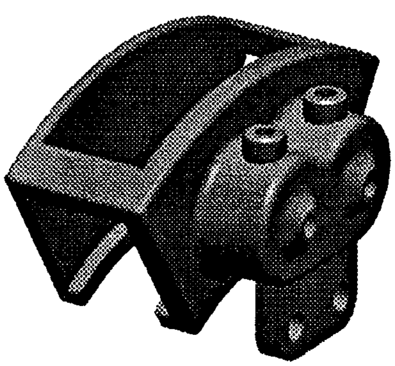

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16. The model is finished.

Storing Solid Geometry Data

The MSC/ARIES Solids software utilizes a hybrid combination of two approaches to storing solid geometry data. These are:

- Constructive Solid Geometry (CSG)
- Boundary Representation (BREP)

Constructive Solid Geometry (CSG)

This method records the *primitives* and *construction operations* that were used to create a complex solid model. The system generates a file, called the *CSG Tree*, that captures the "recipe" of how the model was built. It also allows the recipe to change, enabling solid modifications at the component level.

Additionally, the CSG Tree can be used as a *macro file,* — an executable program which can automate geometry building and, with the incorporation of parametric variables, enables the user to generate "families of parts", objects of similar shapes but varying sizes.

Boundary Representation (BREP)

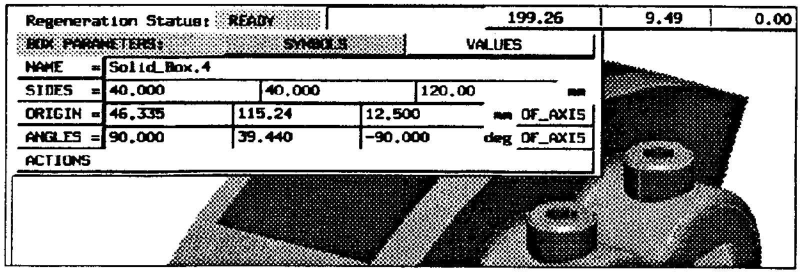
This method stores database information on the *topological entities* (e.g. vertex, edge, face) and *connectivity* relationships that define the boundary of a solild model.

Modifying Solid Models

The MSC/ARIES Parametrics Module provides the ability to change all components of complex solid models (primitives, features, operations) at any time, even if they had been unioned, subtracted, or intersected with other solids.

To modify a solid model, the user points to the portion of the model that is to be changed. The parameter table or the dimensions of the component are displayed and new values can be entered for any of the items.

Let us assume that in our example, the caliper, the size of the cutout must be changed. The cutout was made by subtracting a cutting tool (box) from the base solid (solid of revolution). The parameters of the box can be modified to accommodate the change, even though it has been subtracted and is no longer present on the screen.



Parameter Table of a Box

Similarly, hole parameters, extrusion depths, sweep angles of revolutes, even curve profiles can be changed as needed to optimize the shape of the design.

Feature Suppression

Certain features, while part of the geometric design of a solid model, may not be important for analysis. Small holes, fillets, chamfers that are in low-stress areas may be temporarily *suppressed* via the MSC/ARIES Parametrics application. This eliminates the need to refine a mesh in those unimportant areas and reduces the time required for meshing and analysis. Suppressed features can be re-activated any time.

The Design Rule Processor

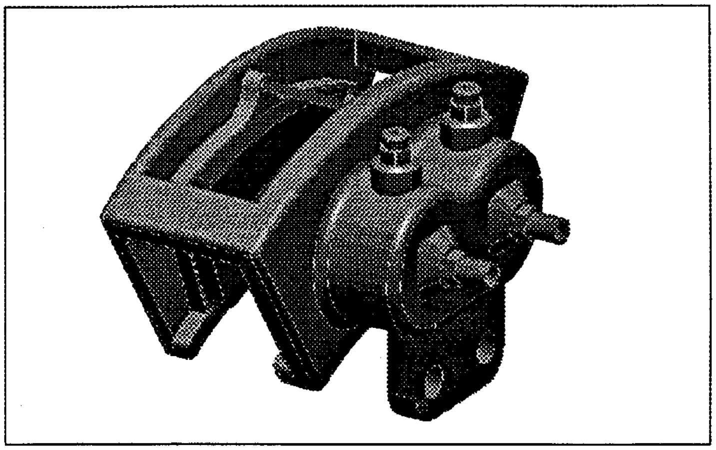
The Design Rule Processor (DRP) enables the design engineer to use engineering rules, in the form of systems of equations, to constrain and control the relationship of features in the solid model. When the values of variables are changed and the equations solved, the solid model is parametrically updated based on the new data.

Solid Models in Assemblies

An assembly is a collection of any number of parts. With a solid model display, the design engineer can easily illustrate the positioning of parts relative to one another in an assembly (especially if each part is made in a different color), and instruct the system to find and highlight possible interferences between parts.

An assembly can be built with the "bottom-up" method, when all parts are first completed and then placed into an assembly, or "top-down", which allows the engineer to design parts in the context of assembly. The latter approach lends itself more to design activities, as one part is designed, other components in the assembly can be viewed and their geometry referenced.

The assembly for which the caliper was created consists of a total of 10 parts. Some of these parts may be standard parts for a company and may be found in a parts library.
If desired, a Bill of Materials can be generated for the assembly to list its components.

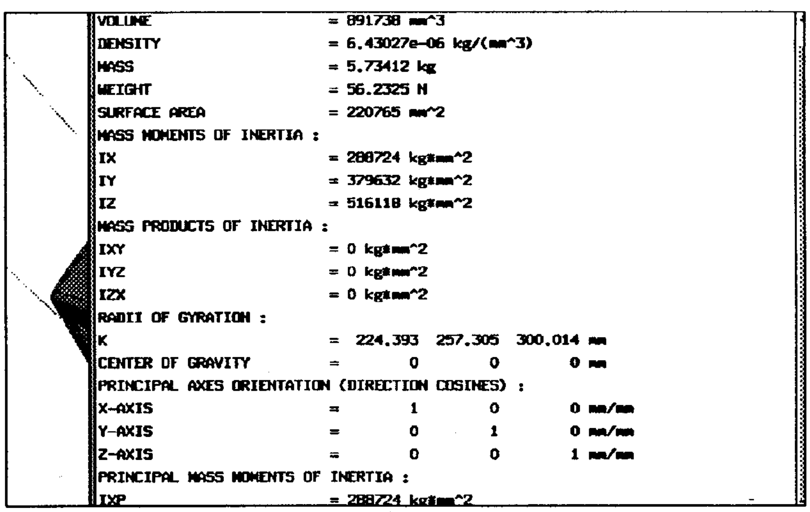


Caliper Assembly

Mass Properties

Parts created with MSC/ARIES Solids can be assigned various material properties. An extensive material library is supplied with the system. Users can generate additional material property definitions by simply filling in values in a ready-made table.

Mass properties can be calculated for individual parts or entire assemblies. Mass- and section properties can be obtained for any design unit definitions in any user-specified coordinate system.



```
VOLUME                    = 891738 mm^3
DENSITY                   = 6.43027e-06 kg/(mm^3)
MASS                      = 5.73412 kg
WEIGHT                    = 56.2325 N
SURFACE AREA              = 220765 mm^2
MASS MOMENTS OF INERTIA :
IX                        = 288724 kg*mm^2
IY                        = 379632 kg*mm^2
IZ                        = 516118 kg*mm^2
MASS PRODUCTS OF INERTIA :
IXY                       = 0 kg*mm^2
IYZ                       = 0 kg*mm^2
IZX                       = 0 kg*mm^2
RADII OF GYRATION :
K                         =  224.393  257.305  300.014 mm
CENTER OF GRAVITY         =        0        0        0 mm
PRINCIPAL AXES ORIENTATION (DIRECTION COSINES) :
X-AXIS                    =        1        0        0 mm/mm
Y-AXIS                    =        0        1        0 mm/mm
Z-AXIS                    =        0        0        1 mm/mm
PRINCIPAL MASS MOMENTS OF INERTIA :
IXP                       = 288724 kg*mm^2
```

Mass Property Output for the Assembly

Creating the Finite Element Model

MSC/ARIES FEM integrates the finite element model with the solid model. The solid geometric model is called up in *ENVIRONMENT* where the applied loads and restraints (boundary conditions) are specified (Note that in MSC/ARIES FEM *constraints* are called "restraints"). Loads and restraints are applied directly on the solid model as *geometric loads and restraints.*

In the *FEM* application the model is meshed using automatic or mapped mesh generation techniques.When the mesh is completed, loads and restraints are converted to their nodal or elemental equivalents.

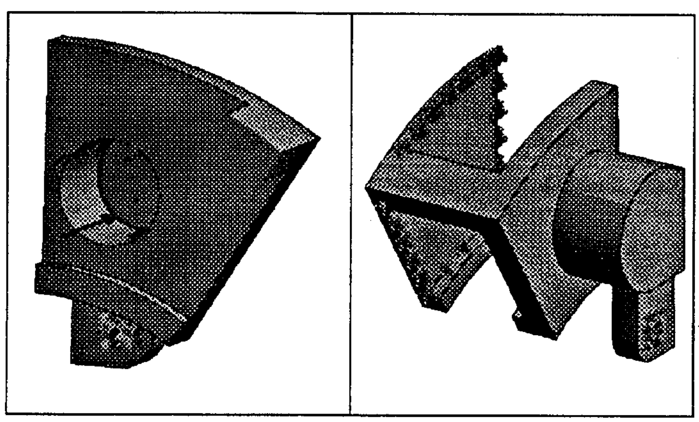
The finite element model is analyzed on-line or submitted for analysis elsewhere. The FEResults application provides analysis results in several different forms, such as visual displays, numerical listings, graphs. Result data can be output on the entire model or information can be obtained on selected elements or gridpoints (in MSC/ARIES FEM gridpoints are called "nodes").

Environment

In this application loads and restraints are defined directly on the solid model. Simplifications can be made to the geometry to reduce the complexity of the finite element model. For example, the user can take advantage of model symmetry or temporarily suppress certain features.

A constant face pressure was applied on the end face of the piston hole. Restraints were applied on the inner face of the caliper and on the inner faces of the mounting holes (a two-view layout and sectioning facilitate access to these faces).

The ENVIRONMENT symbol for face pressure is a set of directed arrows connected by a solid band around the edges of the face on which it is applied. Translational restraint symbols are wedge shaped and rotational restraints are cone shaped. Restraints applied to a face display along the edges of the face.



The Environment Model

FEM

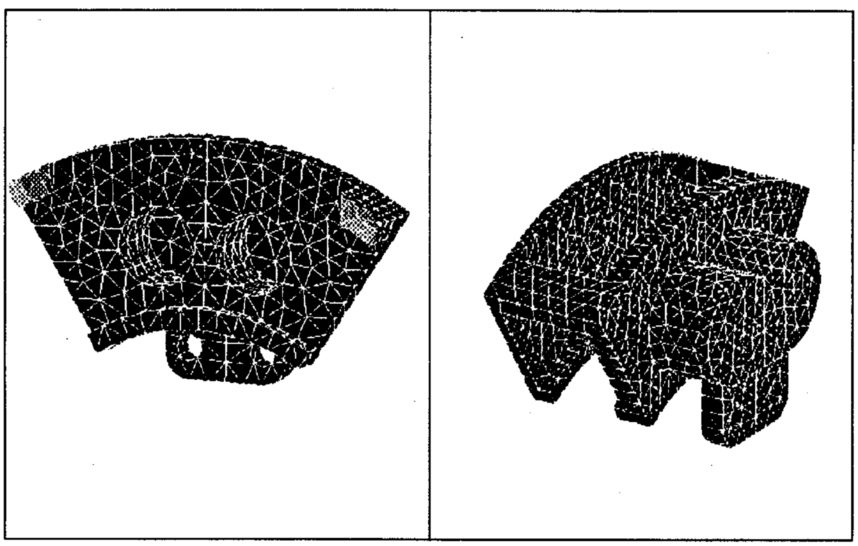
In the Finite Element module the model is first meshed. Meshing can be *automatic* or *mapped.* The automatic mesh generator is capable of meshing complex solid or surface geometry with very little user effort.

Two-dimensional automatic meshes are "quad-dominant"; the mesh consists primarily of quadrilateral elements. Occasionally the mesh will have triangular elements where the part geometry necessitates it.

Three-dimensional automatic meshes are generated with tetrahedral elements. MSC/ARIES recommends that *quadratic elements* (10 -noded TETRAs with midside nodes) be used for three-dimensional solid auto meshing because their accuracy is better than of the linear (4-noded) TETRA elements.

Mapped meshing is an interactive meshing technique that requires that the geometry conform to predefined meshable shapes. Complex solids may have to be subdivided to these required *regions*. Three-dimensional mapped meshes can be made of brick (HEXA) elements which may be linear or quadratic.
All meshes can be translated, rotated, and mirrored.

Once the mesh is done, loads and restraints are converted automatically into nodal or element loads and restraints, and the model is then ready to be submitted for analysis.



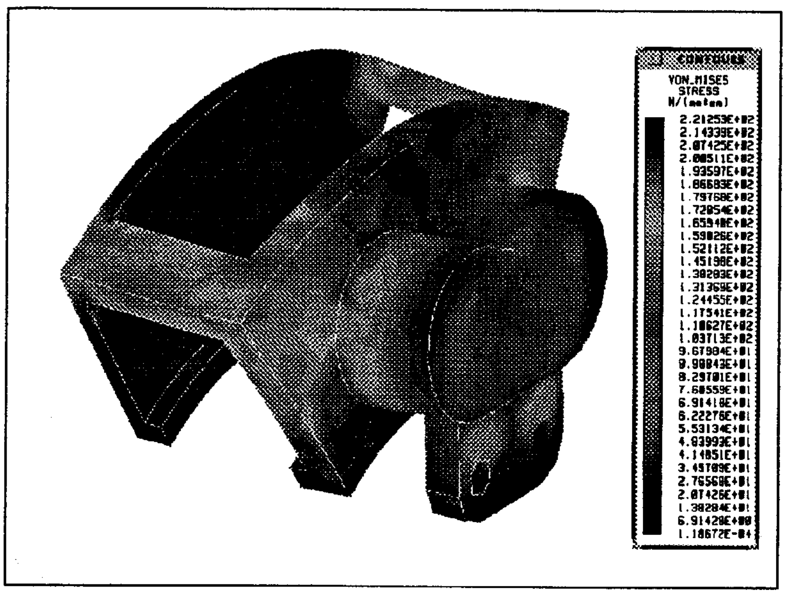
Meshed Finite Element Model

Postprocessing Finite Element Results

The FE_Result application processes the analysis results into visual and numerical displays.

Visual displays include

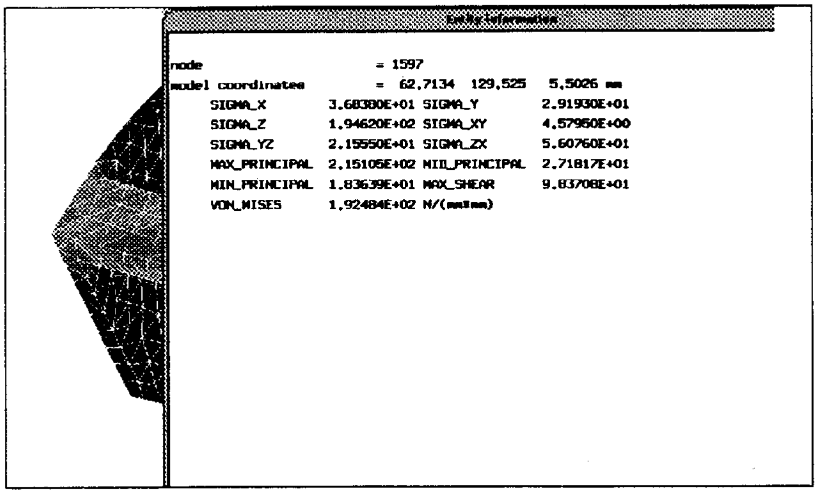
- the deformed shape with user controlled scale factor for displaying deformation.
- contour plots of stresses and displacements, with user-selected stress components, user-defined color schemes and display methods.
- XY-graph of results versus input data.
- highlighting of nodes or elements according to user-selected criteria.
- animation of deformed shapes.



Analysis Results Display

Numerical displays consist of

- Summary on all analysis result data.
- Analysis results for user-selected nodes or elements.
- Combination of results from several analyses.
- Error calculation.



Nodal Information Output

Summary

The MSC/ARIES Solids, MSC/ARIES Parametrics, and MSC/ARIES FEM software modules are powerful tools that allow mechanical engineers to test their designs at the software level using solid modeling, parametric design change, and analysis capabilities.

Geometric solid models are easily transitioned into Finite Element models, with the application of loads and restraints and the generation of meshes.

Analysis results can be displayed and evaluated.

The combination and integration of solid modeling, analysis, and post-processing provides the means of developing *software prototypes,* — the evaluation of the design at the software level.