

Chapter I Introduction

- ◇ SAP2000 is the latest and most powerful version of the well-known SAP series of structural analysis programs.

About This Manual

- ◇ This manual describes the basic and most commonly used modeling and analysis features offered by the SAP2000 structural analysis program. It is imperative that you read this manual and understand the assumptions and procedures used by the program before attempting to create a model or perform an analysis.
- ◇ The complete set of modeling and analysis features is described in the *SAP2000 Analysis Reference*.
- ◇ As background material, you should first read chapter “The Structural Model” in the *SAP2000 Getting Started* manual earlier in this volume. It describes the overall features of a SAP2000 model. The present manual will provide more detail on some of the elements, properties, loads, and analysis types.

Topics

- ◇ Each chapter of this manual is divided into topics and subtopics. Most chapters

第一章 简介

- ◇ SAP2000 是著名的 SAP 系列结构分析程序的最新版本。

关于本书

- ◇ 本书介绍 SAP2000 结构分析程序最常用的建模和分析功能。在进行建模和分析之前，请务必阅读本书，理解其基本假定和分析步骤。
- ◇ 详细的建模和分析功能，请参阅《SAP2000 分析指南》。
- ◇ 作为背景知识，用户应该首先了解《SAP2000 入门教程》的‘结构模型’部分，其描述了 SAP2000 结构模型的各项特性。本书主要介绍常用的单元、属性、荷载和分析类型。

主题

- ◇ 本书各章均划分为多个主题和子主题。大部分章节

begin with a list of topics covered. Following the list of topics is an Overview which provides a summary of the chapter.

在开始处列出本章的所有主题，主题之后是概括本章内容的综述。

Bibliographic References

参考文献

- ◇ References are indicated throughout this manual by giving the name of the author (s) and the date of publication, using parentheses. For example:

See Wilson and Tetsuji (1983).

It has been demonstrated (Wilson, Yuan, and Dickens, 1982) that...

- ◇ 本书指明引用的参考文献，注明作者的姓名和出版日期，用括号表示。如：

参见 Wilson and Tetsuji (1983)

... .. 已被证明 (Wilson, Yuan and Dickens, 1982)

- ◇ All bibliographic references are listed in alphabetical order in Chapter “Bibliography”.

- ◇ 所有参考文献均在“参考文献”一章中按字母顺序列出。

Chapter II Objects and Elements

第二章 对象和单元

- ◇ The physical structural members in a SAP2000 model are represented by objects. Using the graphical user interface, you “draw” the geometry of an object, then “assign” properties and loads to the object to completely define the model of the physical member.
- ◇ The following object types are available, listed in order of geometrical dimension:
 - **Point objects**, of two types:
 - ✓ **Joint objects:** These are automatically created at the corners or ends of all other types of objects below, and they can be explicitly added to model supports or other localized behavior.
 - ✓ **Grounded (one-joint) link objects:** Used to model special support behavior such as isolators, dampers, gaps, multilinear springs, and more. These are not covered in this manual.
 - **Line objects**, of two types
 - ✓ **Frame/cable objects:** Used to model beams, columns, braces, trusses, and/or cable members
- ◇ SAP2000模型的实际结构构件用对象来体现。通过图形界面 用户首先“画出”对象的几何特征，然后“指定”荷载和属性到对象上，完成实际构件模型。
- ◇ 程序中包含下列对象类型，以几何顺序列出：
 - 点对象，包含两类：
 - ✓ 节点对象：自动在以下所有对象的角部或端部生成，可以明确地加入支座或其他局部特性。
 - ✓ 对地（单点）连接对象：用来模拟特殊支承特性，如隔离器、阻尼器、缝隙、多线性弹簧等。本书不包含这些内容。
 - 线对象，包含两类：
 - ✓ 杆件/索对象用来模拟梁、柱、支撑桁架和索等。

- ✓ **Connecting (two-joint) link objects:** Used to model special member behavior such as isolators, dampers, gaps, multilinear springs, and more. Unlike frame/cable objects, connecting link objects can have zero length. These are not covered in this manual.
- **Area objects:** Used to model walls, floors, and other thin-walled members, as well as two-dimensional solids (plane stress, plane strain, and axisymmetric solids). Only shell-type area objects are covered in this manual
- **Solid objects:** Used to model three-dimensional solids. These are not covered in this manual.
- ◇ As a general rule, the geometry of the object should correspond to that of the physical member. This simplifies the visualization of the model and helps with the design process.
- ◇ If you have experience using traditional finite element programs, including earlier versions of SAP2000, you are probably used to meshing physical models into smaller finite elements for analysis purposes. Object-based modeling largely eliminates the need for doing this.
- ✓ 连接 (两点) 对象: 用来模拟特殊构件特性 如隔离器、阻尼器、缝隙、多线性弹簧等。与杆件 / 索对象不同, 连接对象可以是零长度。本书不包含这些内容。
- 面对象: 用来模拟墙、楼板及其他薄壁构件, 也可模拟二维实体 (平面应力、平面应变、轴对称实体)。本书只包含壳单元。
- 实体对象: 用来模拟三维实体。本书不包含这些内容。
- 作为一般原则, 对象的几何特性应与实际构件一致。这可增强模型的可视性并有利于设计过程。
- 如果你有使用传统有限元程序的经验, 包括早期版本的 SAP2000, 可能已习惯于把物理模型划分为更小的单元进行分析。基于对象建模大大消除了这种必要。

- ◇ For users who are new to finite-element modeling, the object-based concept should seem perfectly natural.
- ◇ When you run an analysis, SAP2000 automatically converts your object-based model into an element-based model that is used for analysis. This element-based model is called the analysis model, and it consists of traditional finite elements and joints (nodes). Results of the analysis are reported back on the object-based model.
- ◇ You have control over how the meshing is performed, such as the degree of refinement, and how to handle the connections between intersecting objects. You also have the option to manually mesh the model, resulting in a one-to-one correspondence between objects and elements.
- ◇ In this manual, the term “element” will be used more often than “object”, since what is described herein is the finite-element analysis portion of the program that operates on the element-based analysis model. However, it should be clear that the properties described here for elements are actually assigned in the interface to the objects, and the conversion to analysis elements is automatic.
- ◇ 对有限元建模的初学者来说, 基于对象的概念更加自然些。
- ◇ 当进行分析时, SAP2000 自动将基于对象的模型转换成基于单元的模型进行分析。这种基于单元的模型称为分析模型, 它包含传统有限元的单元和节点。分析的结果会自动转换到基于对象的模型上。
- ◇ 用户可以控制网格划分, 如自由度细分、相交对象的连接处理等。也可以手工划分模型网格, 使对象和单元一一对应。
- ◇ 本书中“单元”比“对象”使用得更多, 因为本书介绍的是程序的有限元分析部分, 其处理过程基于单元的分析模型。然而, 需要明确的是, 这里描述的单元属性在程序界面中是指定给对象的, 由程序自动转换到单元上。

Chapter III Coordinate Systems

- ◇ Each structure may use many different coordinate systems to describe the location of points and the directions of loads, displacement, internal forces, and stresses. Understanding these different coordinate systems is crucial to being able to properly define the model and interpret the results.

Topics

- Overview
- Global Coordinate System
- Upward and Horizontal Directions
- Local Coordinate Systems

Overview

- ◇ Coordinate systems are used to locate different parts of the structural model and to define the directions of loads, displacements, internal forces, and stresses.
- ◇ All coordinate systems in the model are defined with respect to a single, global X-Y-Z coordinate system. Each part of the model (joint, element, or constraint) has its own local 1-2-3 coordinate system. In addition, you may create alternate coordinate systems that are used to define locations and directions. All coordinate systems are three-dimensional, right-handed, rectangular (Cartesian) systems.

第三章 坐标系

- ◇ 每个结构可能使用多个不同的坐标系来描述点的位置和荷载、位移、内力及应力的方向。理解这些不同的坐标系对正确建模和解释计算结果是至关重要的。

主题

- 概述
- 全局坐标系
- 向上和水平方向
- 局部坐标系

概述

- ◇ 坐标系用来定位结构模型的不同部分和定义荷载、位移、内力和应力的方向。
- ◇ 模型中的所有坐标系均参照一个单一的全局 X-Y-Z 坐标系定义。模型的各个部分（节点、单元或限制）都有各自的 1-2-3 局部坐标系。另外，也可以创建其他坐标系来定义位置和方向。所有坐标系都是三维右手直角坐标系（笛卡尔坐标系）。

- ◇ SAP2000 always assumes that Z is the vertical axis, with + Z being upward. The upward direction is used to help define local coordinate systems, although local coordinate systems themselves do not have an upward direction.
- ◇ For more information and additional features, see Chapter “Coordinate Systems” in the *SAP2000 Analysis Reference* and the Help Menu in the SAP2000 graphical user interface.
- ◇ SAP2000 总是假设 Z 为竖直方向, + Z 向上。尽管局部坐标系本身没有向上方向 但向上的 Z 方向可用来帮助定义局部坐标系。
- ◇ 更多的信息和特性请参见《SAP2000分析指南》中的“坐标系”一章及 SAP2000 图形用户界面的帮助菜单。

Global Coordinate System

全局坐标系

- ◇ The **global coordinate system** is a three-dimensional, right-handed, rectangular coordinate system. The three axes, denoted X, Y, and Z, are mutually perpendicular and satisfy the right-hand rule. The location and orientation of the global system are arbitrary.
- ◇ Locations in the global coordinate system can be specified using the variables x , y , and z . A vector in the global coordinate system can be specified by giving the locations of two points, a pair of angles, or by specifying a coordinate direction. Coordinate directions are indicated using the values X, Y, and Z. For example, + X defines a vector parallel to and directed along the positive X axis. The sign is required.
- ◇ All other coordinate systems in the model are defined with respect to the global coordinate system.
- ◇ 全局坐标系是右手三维直角坐标系 记为 X、Y、Z。三个方向轴相互垂直, 满足右手法则。全局坐标系的位置和方位任意。
- ◇ 全局坐标系中的位置用变量 x 、 y 、 z 表示。全局坐标系中的矢量可用两点的位置、一对角度或一个坐标方向来确定。坐标方向用 X、Y、Z 指定 例如, + X 表示一个平行于 X 轴且指向正 X 方向的矢量。这里正负号是必须的。
- ◇ 模型中的其他坐标系都是参照全局坐标系定义的。

Upward and Horizontal Directions

- ◇ SAP2000 always assumes that Z is the vertical axis, with $+Z$ being upward. Local coordinate systems for joints, elements, and ground-acceleration loading are defined with respect to this upward direction. Self-weight loading always acts downward, in the $-Z$ direction.
- ◇ The X - Y plane is horizontal. The primary horizontal direction is $+X$. Angles in the horizontal plane are measured from the positive half of the X axis, with positive angles appearing counter-clockwise when you are looking down at the X - Y plane.

向上和水平方向

- ◇ SAP2000 总是假设 Z 为竖直方向, $+Z$ 向上。节点、单元、地面加速度荷载的局部坐标系均相对于该向上的方向定义。自重荷载总是向下, 即 $-Z$ 方向。
- ◇ X - Y 平面是水平面, 水平主方向为 $+X$ 。水平面内的角度从 X 轴的正半轴度量。从 $+Z$ 向下看 X - Y 平面, 逆时针角度为正。

Local Coordinate Systems

- ◇ Each part (joint, element, or constraint) of the structural model has its own local coordinate system used to define the properties, loads, and response for that part. The axes of the local coordinate systems are denoted 1, 2, and 3. In general, the local coordinate systems may vary from joint to joint, element to element, and constraint to constraint.
- ◇ There is no preferred upward direction for a local coordinate system. However, the joint and element local coordinate systems are defined with respect to the global upward direction, $+Z$.

局部坐标系

- ◇ 结构模型的各部分(节点、单元或限制)均有自己的局部坐标系, 用来定义其属性、荷载和响应。局部坐标系用 1、2、3 表示。通常, 对不同的节点、单元和限制来说, 其局部坐标系是不同的。
- ◇ 局部坐标系没有向上主方向。但是, 节点和单元的局部坐标系是参照全局坐标系的 $+Z$ 方向定义的。

- ◇ The joint local 1-2-3 coordinate system is normally the same as the global X-Y-Z coordinate system.
- ◇ For the Frame and Shell elements, one of the element local axes is determined by the geometry of the individual element. You may define the orientation of the remaining two axes by specifying a single angle of rotation.
- ◇ The local coordinate system for a Diaphragm Constraint is normally determined automatically from the geometry or mass distribution of the constraint. Optionally, you may specify one global axis that determines the plane of a Diaphragm Constraint; the remaining two axes are determined automatically.
- ◇ For more information:
 - See Topic “Local Coordinate System” in Chapter “The Frame Element.”
 - See Topic “Local Coordinate System” in Chapter “The Shell Element.”
 - See Topic “Local Coordinate System” in Chapter “Joints and Degrees of Freedom.”
 - See Topic “Diaphragm Constraint” in Chapter “Joint Constraints.”
- ◇ 节点的默认 1-2-3 局部坐标系与全局坐标系 X-Y-Z 相同(注 原点在节点上)
- ◇ 对杆件和壳单元来说,其某一局部坐标轴由单元自身的几何特性确定。用户可以指定一个旋转角度来定义其余两个坐标轴的方位。
- ◇ 薄膜限制(注:面内位移限制)的局部坐标系通常由该限制的几何性质或质量分布自动确定。另外,用户可以指定一个全局坐标轴来定义薄膜限制平面,其余二轴自动确定。
- ◇ 更多内容请参见:
 - “杆件单元”一章的“局部坐标系”主题。
 - “壳单元”一章的“局部坐标系”主题。
 - “节点和自由度”一章的“局部坐标系”主题。
 - “节点限制”一章的“薄膜限制”主题。

Chapter IV The Frame Element

- ◇ The **Frame element** is used to model beam-column and truss behavior in planar and three-dimensional structures. The frame element can also be used to model cable behavior when nonlinear properties are added (e. g., tension only, large deflections). Although everything described in this chapter can apply to cables, cable-specific behavior is not discussed.

Topics

- Overview
- Joint Connectivity
- Degrees of Freedom
- Local Coordinate System
- Section Properties
- Insertion Point
- End Offsets
- End Releases
- Mass
- Self-Weight Load
- Concentrated Span Load
- Distributed Span Load
- Internal Force Output

Overview

- ◇ The Frame element uses a general, three dimensional, beam-column formulation which includes the effects of biaxial bending, torsion, axial deformation, and biaxial shear deformations. See Bathe and Wilson (1976).

第四章 杆件单元

- ◇ 杆件单元用于模拟平面或空间的梁、柱和桁架行为。当加入非线性特性时（如仅拉、大变形），杆件单元也可用于模拟索特性。尽管本章的讨论内容都适用于索，但索特性不在本章讨论。

主题

- 概述
- 节点连接
- 自由度
- 局部坐标系
- 截面特性
- 插入点
- 端部偏移
- 端部释放
- 质量
- 自重荷载
- 跨内集中荷载
- 跨内分布荷载
- 内力输出

概述

- ◇ 杆件单元使用通用的三维梁柱公式，包含了双向弯曲、扭转、轴向变形、双向剪切等效应。参见 Bathe and Wilson(1976)。

- ◇ Structures that can be modeled with this element include:
 - Three-dimensional frames
 - Three-dimensional trusses
 - Planar frames
 - Planar grillages
 - Planar trusses
 - Cables
 - ◇ A Frame element is modeled as a straight line connecting two points. In the graphical user interface, you can divide curved objects into multiple straight objects, subject to your specification.
 - ◇ Each element has its own local coordinate system for defining section properties and loads, and for interpreting output.
 - ◇ Each Frame element may be loaded by self-weight, multiple concentrated loads, and multiple distributed loads.
 - ◇ Insertion points and end offsets are available to account for the finite size of beam and column intersections. End releases are also available to model different fixity conditions at the ends of the element.
 - ◇ Element internal forces are produced at the ends of each element and at a user-specified number of equally-spaced output stations along the length of the element.
- 可用杆件单元模拟的结构有：
- 三维框架
 - 三维桁架
 - 平面框架
 - 平面网架
 - 平面桁架
 - 悬索
- 杆件单元模型为连接两点的一条直线。根据用户指定，可在用户界面中将曲线对象划分为多条直线
- 每个杆件单元有自身的局部坐标系来定义截面特性、荷载和解释输出结果。
- 每个杆件单元可施加自重、多个集中力和多个分布荷载。
- 使用插入点和端部偏移可考虑梁柱交点处的尺寸。使用端部释放可模拟单元端部的不同连接条件（注：如铰接、滑动、刚接等）。
- 单元内力在杆端和在用户指定的沿杆长等间距分布的位置输出。

- ◇ Cable behavior is modeled using the frame element and adding the appropriate features. You can release the moments at the ends of the elements, although we recommend that you retain small, realistic bending stiffness instead. You can also add nonlinear behavior as needed, such as the no-compression property, tension stiffening (P-Delta effects), and large deflections. These features require nonlinear analysis, and are not covered in this manual.
- ◇ For more information and additional features, see Chapter “The Frame Element” in the *SAP2000 Analysis Reference*.
- ◇ 增加相应的特性后, 可用杆件单元模拟悬索特性。用户可释放单元端部弯矩, 但建议保留很小的真实抗弯刚度。也可按需添加非线性特性, 如不可压缩特性、拉伸刚化 (P-Delta 效应)、大变形等。这些特性要求进行非线性分析, 不在本书讨论。
- ◇ 更多的信息和特性, 参见《SAP2000 分析指南》的“杆件单元”一章。

Joint Connectivity

- ◇ A Frame element is represented by a straight line connecting two joints, i and j , unless modified by joint offsets as described below. The two joints must not share the same location in space. The two ends of the element are denoted end I and end J, respectively.
- ◇ By default, the neutral axis of the element runs along the line connecting the two joints. However, you can change this using the insertion point, as described in Topic “Insertion Point”.
- ◇ 除非按下述节点偏移方法修改杆件单元用连接 i 和 j 两点的直线表示。该两点不能在同一空间位置, 单元两端分别记为 I 和 J。
- ◇ 默认情况下, 单元中性轴为连接两点的直线。但可用插入点改变中性轴, 参见主题“插入点”所述。

Joint Offsets

- ◇ Sometimes the axis of the element can not be conveniently specified by joints
- ◇ 有时, 不能方便地用连接于结构上其他单元的节点指

节点连接

节点偏移

that connect to other elements in the structure. You have the option to specify **joint offsets** independently at each end of the element. These are given as the three distance components (X, Y, and Z) parallel to the global axes, measured from the joint to the end of the element (at the insertion point.)

- ✧ The two locations given by the coordinates of joints i and j, plus the corresponding joint offsets, define the axis of the element. These two locations must not be coincident. It is generally recommended that the offsets be perpendicular to the axis of the element, although this is not required.
 - ✧ Offsets along the axis of the element are usually specified using end offsets rather than joint offsets. See topic “End Offsets”. End offsets are part of the length of the element, have element properties and loads, and may or may not be rigid. Joint offsets are external to the element, and do not have any mass or loads. Internally the program creates a fully rigid constraint along the joints offsets.
 - ✧ Joint offsets are specified along with the cardinal point as part of the insertion point assignment, even though they are independent features.
 - ✧ For more information:
 - See Topic “Insertion Point” in this chapter.
- ✧ 由节点 i 和 j 的坐标位置，加上相应的节点偏移，定义了单元轴。当然，两个节点的坐标位置不能相同。通常，建议节点偏移垂直于单元轴，但这不是必须的。
 - ✧ 单元沿轴向的偏移通常用单元端部偏移指定，而非用节点偏移。参见主题“端部偏移”。端部偏移是单元长度的一部分，具有单元特性和荷载，可为刚性，也可不是。节点偏移在单元的外部，没有任何质量或荷载。程序在内部沿节点偏移创建了一个全刚性限制。
 - ✧ 节点偏移和控制点一起被指定为插入点的一部分，尽管它们有独立的特性。
 - ✧ 更多信息参见
 - 本章“插入点”主题

定单元轴。用户可选择在单元的各端独立定义节点偏移，其由三个平行于全局坐标轴的距离分量（X、Y、Z）给出，即从节点到单元端部（插入点处）的距离分量。

- See Topic “End Offsets” in this chapter.

Degrees of Freedom

- ◇ The Frame element activates all six degrees of freedom at both of its connected joints. If you want to model truss or cable elements that do not transmit moments at the ends, you may either:
 - Set the geometric Section properties j , $i33$, and $i22$ all to zero (a is non-zero; $as2$ and $as3$ are arbitrary), or
 - Release both bending rotations, $R2$ and $R3$, at both ends and release the torsional rotation, $R1$, at either end.
- ◇ For more information:
 - See Topic “Degrees of Freedom” in Chapter “Joints and Degrees of Freedom.”
 - See Topic “Section Properties” in this chapter.
 - See Topic “End Releases” in this chapter.

Local Coordinate System

- ◇ Each Frame element has its own element **local coordinate system** used to define section properties, loads and output. The axes of this local system are denoted 1, 2 and 3. The first axis is directed along the length of the element; the remaining two axes lie in the plane perpendicular to the element with an orientation that you specify.

- 本章“端部偏移”主题

自由度

- ◇ 杆件单元两端节点的六个自由度都被激活。若需模拟不传递弯矩的桁架或悬索 可以选择下述方法：
 - 令几何截面特性 j 、 $i33$ 和 $i22$ 为零 (a 为非零, $as2$ 和 $as3$ 任意)；
 - 释放两端两个方向的弯曲转动 $R2$ 和 $R3$ 在任意端释放扭转转动 $R1$ 。
- ◇ 更多信息参见
 - “节点和自由度”一章的“自由度”主题
 - 本章“截面特性”主题
 - 本章“端部释放”主题

局部坐标系

- ◇ 每个杆件单元有其自身的局部坐标系，用来定义截面特性、荷载和输出。局部坐标系的主轴用 1、2 和 3 表示，1 轴方向指向单元的长度方向，其余的两个主轴位于垂直于单元的平面内，方位由用户指定。

- ✧ It is important that you clearly understand the definition of the element local 1-2-3 coordinate system and its relationship to the global X-Y-Z coordinate system. Both systems are right-handed coordinate systems. It is up to you to define local systems which simplify data input and interpretation of results.
- ✧ In most structures the definition of the element local coordinate system is extremely simple using the **default orientation** and the **Frame element coordinate angle**. Additional methods are available.
- ✧ For more information:
 - See Chapter “Coordinate Systems” for a description of the concepts and terminology used in this topic.
 - See Topic “Advanced Local Coordinate System” in Chapter “The Frame Element” in the *SAP2000 Analysis Reference*.
 - See Topic “Joint Offsets” in this chapter.
- ✧ 理解单元局部坐标系的定义及其与全局坐标系的关系非常重要。两个坐标系都遵循右手法则，用户定义局部坐标系来简化数据输入和解释输出结果。
- ✧ 对大多数结构来说，通过使用程序默认方位和杆件单元坐标角，单元局部坐标系的定义极其简单。当然还有其他方法。
- ✧ 更多的信息请参见
 - “坐标系”一章中关于该主题的概念和术语描述；
 - 《SAP2000 分析指南》中“杆件单元”一章中“高级局部坐标系统”主题。
 - 本章“节点偏移”主题。

Longitudinal Axis 1

- ✧ Local axis 1 is always the longitudinal axis of the element, the positive direction being directed from end I to end J.
- ✧ Specifically, end I is joint i plus its joint offsets (if any), and end J is joint j plus its joint offsets (if any). The axis is determined independently of the cardinal point; see Topic “Insertion Point”.
- ✧ 局部坐标轴 1 总是沿单元纵轴方向，从端部 I 指向 J 为正。
- ✧ 特定地，端部 I 是节点 i 加上其节点偏移量（如果有），端部 J 是节点 j 加上其节点偏移量（如果有）。纵轴独立于控制点定义，参见“插入点”主题。

纵向主轴 1

Default Orientation

- ✧ The default orientation of the local 2 and 3 axes is determined by the relationship between the local 1 axis and the global Z axis:
 - The local 1-2 plane is taken to be vertical, i. e., parallel to the Z axis
 - The local 2 axis is taken to have an upward (+ Z) sense unless the element is vertical, in which case the local 2 axis is taken to be horizontal along the global + X direction
 - The local 3 axis is always horizontal, i. e., it lies in the X-Y plane
- ✧ An element is considered to be vertical if the sine of the angle between the local 1 axis and the Z axis is less than 10^{-3} .
- ✧ The local 2 axis makes the same angle with the vertical axis as the local 1 axis makes with the horizontal plane. This means that the local 2 axis points vertically upward for horizontal elements.

Coordinate Angle

- ✧ The Frame element coordinate angle, **ang**, is used to define element orientations that are different from the default orientation. It is the angle through which the local 2 and 3 axes are rotated about the positive local 1 axis from the default orientation. The rotation for a positive value of **ang** appears counter-clockwise when the local + 1 axis is pointing toward you.

默认方位

- ✧ 局部坐标 2、3 轴的默认方位由局部坐标轴 1 和全局坐标轴 Z 的关系确定：
 - 局部坐标 1-2 平面默认为竖向，平行于主轴 Z。
 - 局部坐标轴 2 默认为朝 + Z 方位。对竖直单元，局部坐标轴 2 默认为 + X 方向。
 - 局部坐标轴 3 总是水平的 即位于 X-Y 平面内。
(注 平行于 X-Y 平面)
- ✧ 如果局部坐标轴 1 和轴 Z 之间夹角的正弦值小于 10^{-3} 则该单元被视为竖向单元；
- ✧ 局部坐标轴 2 与竖轴的夹角等于局部坐标轴 1 与水平面之间的夹角，这也表明水平单元的局部坐标轴 2 竖直向上。

坐标角

- ✧ 杆件单元的坐标角 **ang** 用来定义不同于默认方位的单元方位。坐标角是使局部坐标轴 2 和 3 从默认方位绕轴 1 转向指定方位的转角。当 + 1 指向用户时，正角度值 **ang** 为逆时针转动。

- ✧ For vertical elements, **ang** is the angle between the local 2 axis and the horizontal + X axis. Otherwise, **ang** is the angle between the local 2 axis and the vertical plane containing the local 1 axis. See Figure 4-1 for examples.
- ✧ 对于竖直单元, **ang** 是局部坐标轴 2 和水平主轴 + X 的夹角。除此以外, **ang** 是局部坐标轴 2 与包含局部坐标轴 1 的竖平面之间的夹角。参见图 4-1。

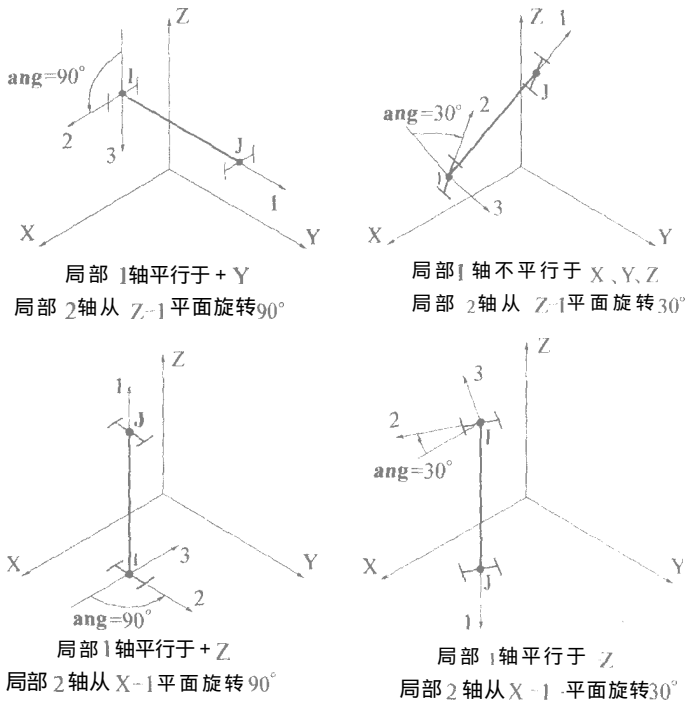


图 4-1 杆件单元坐标角

Section Properties

- ✧ A **Frame Section** is a set of material and geometric properties that describe the cross-section of one or more Frame elements. Sections are defined independently of the Frame elements, and are assigned to the elements.

截面特性

- ✧ 杆件截面是描述一个或多个杆件单元横截面的一组材料和几何特性。杆件截面与杆件单元独立定义, 然后再指定给单元