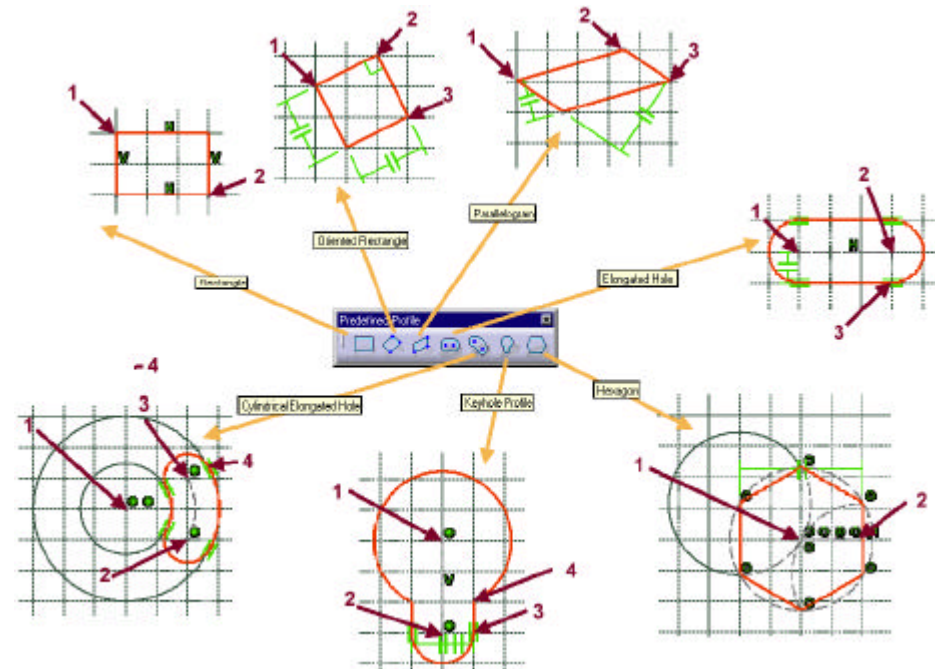


Treinamento *CATIA V5*



Sketcher

BÁSICO



Sketcher Básico

Objetivos do Curso

O objetivo do curso é familiarizar o novo usuário CATIA V5 com o sketcher como uma ferramenta para criar perfis 2D. Estes perfis podem ser usados para gerar tanto sólidos quanto superfícies.

Público Alvo

Novos usuários de CATIA V5

TECMES

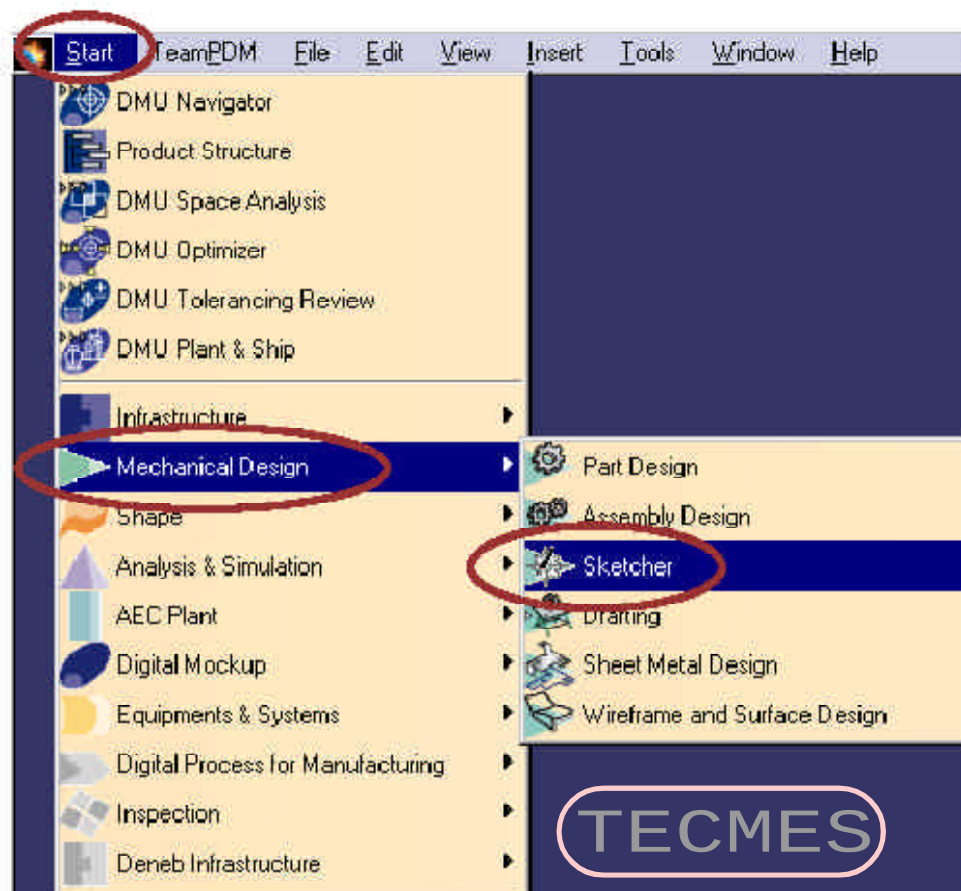
ÍNDICE

Introdução ao Sketcher	3
Desenhando Perfis Simples	8
Modo de trabalho do Sketcher	9
Criando Perfis (Profile)	10
Criando Pontos	11
Criando Linhas	13
Criando Splines	16
Criando Círculos e Arcos	17
Criando Cônicas	18
Criando Eixos	19
Criando Perfis Pré-Definidos	20
Editando Perfis	21
Modificando Perfis	22
Filletts e Chanfros	23
Trim e Break	24
Quick Trim	25
Close	26
Complement	27
Transformações	28
Offset	29
Restrições (constraints)	31
Criando restrições (constraints)	32
Modificando Restrições (constraints)	35
Criando restrições Automaticamente (auto constraint)	36
Animando Restrições (constraints)	37
Relacionando Restrições (constraints)	38
Integração com o 3D	39

Introdução ao Sketcher

Acessando o Workbench

Selecione **Start > Mechanical Design > Sketcher**
depois selecione um plano ou uma face em um objeto

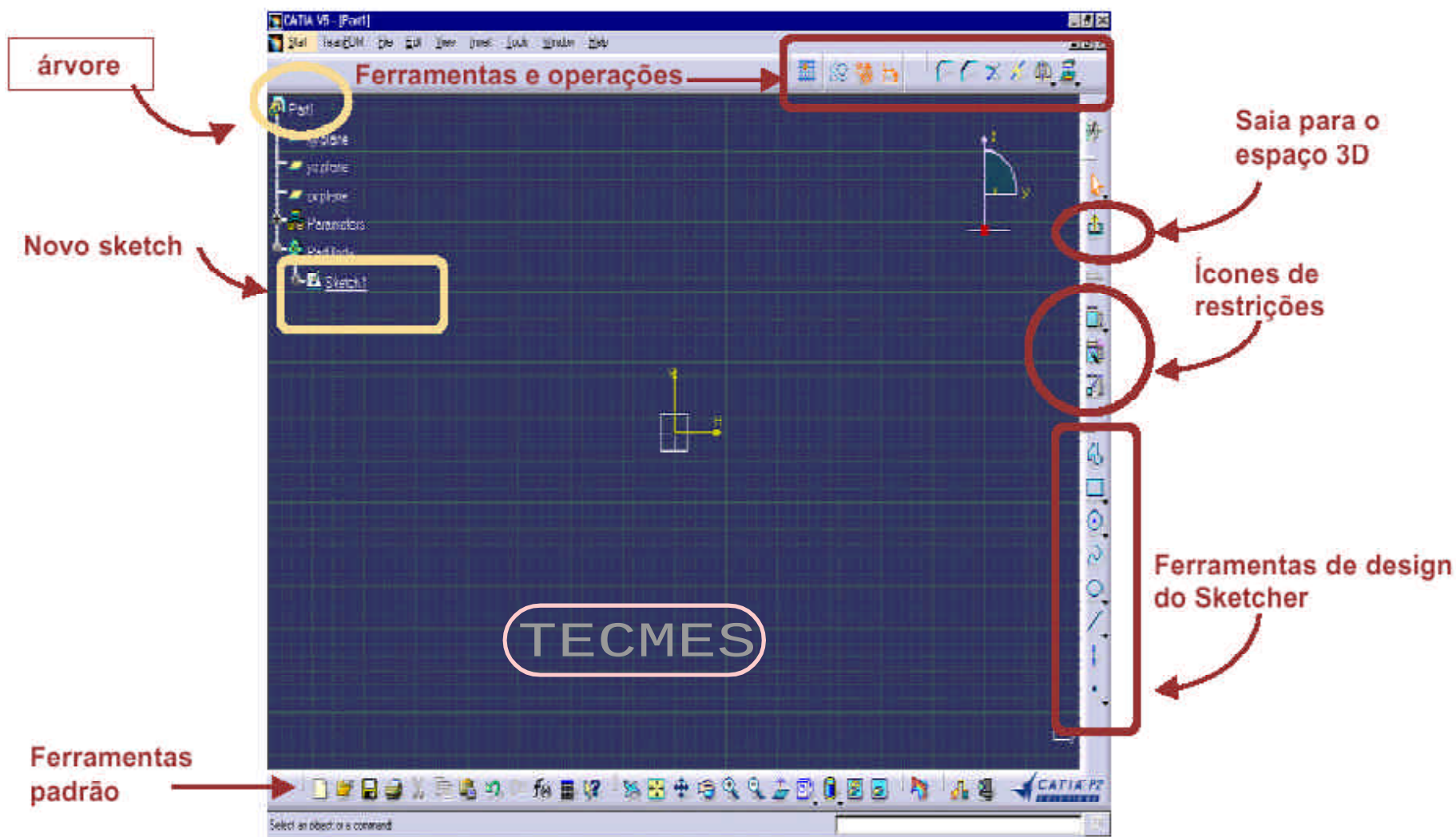


Você pode também acessar o Sketcher
selecionando o ícone Sketcher em qualquer
Workbench onde é possível fazer o sketch

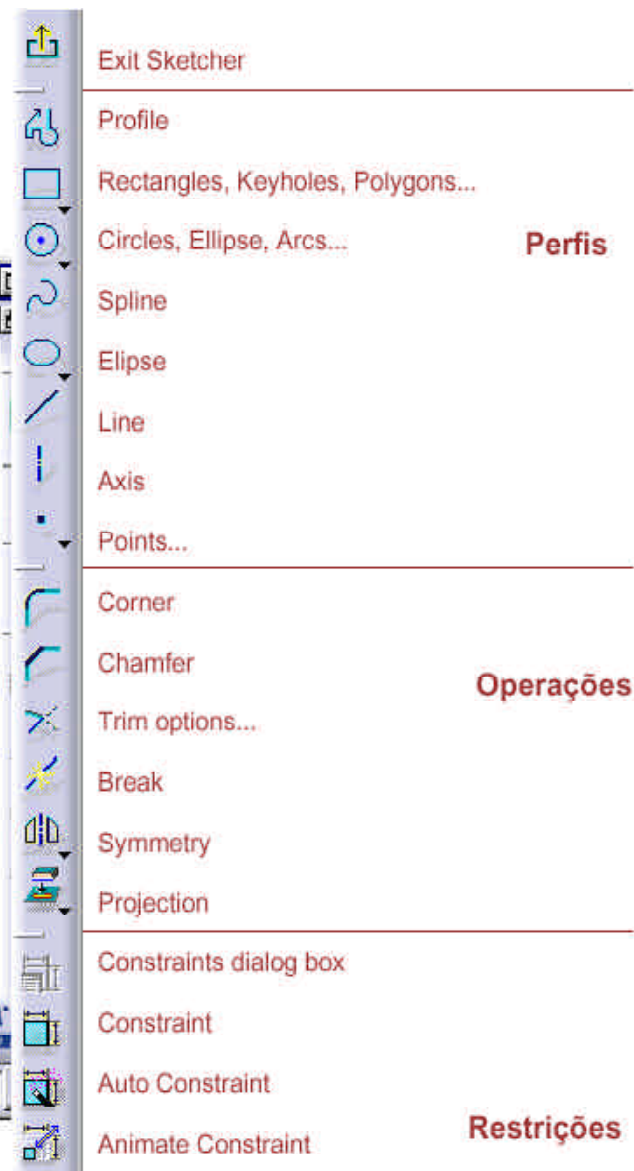
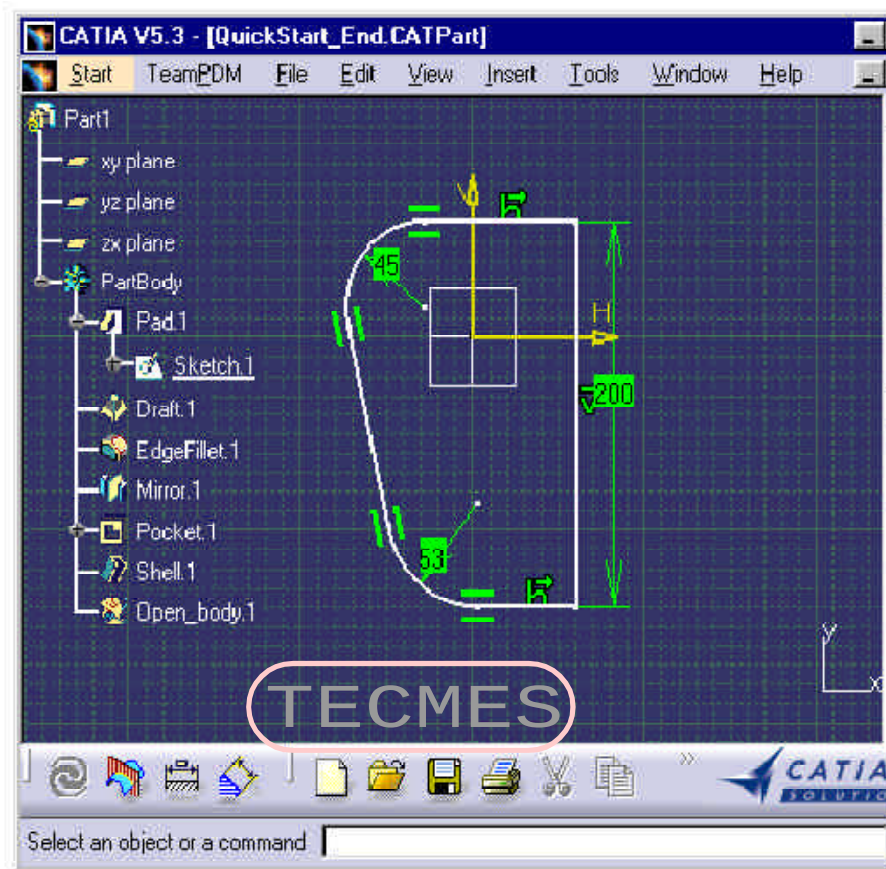


A Interface usuário (1/4): Sketcher Workbench geral

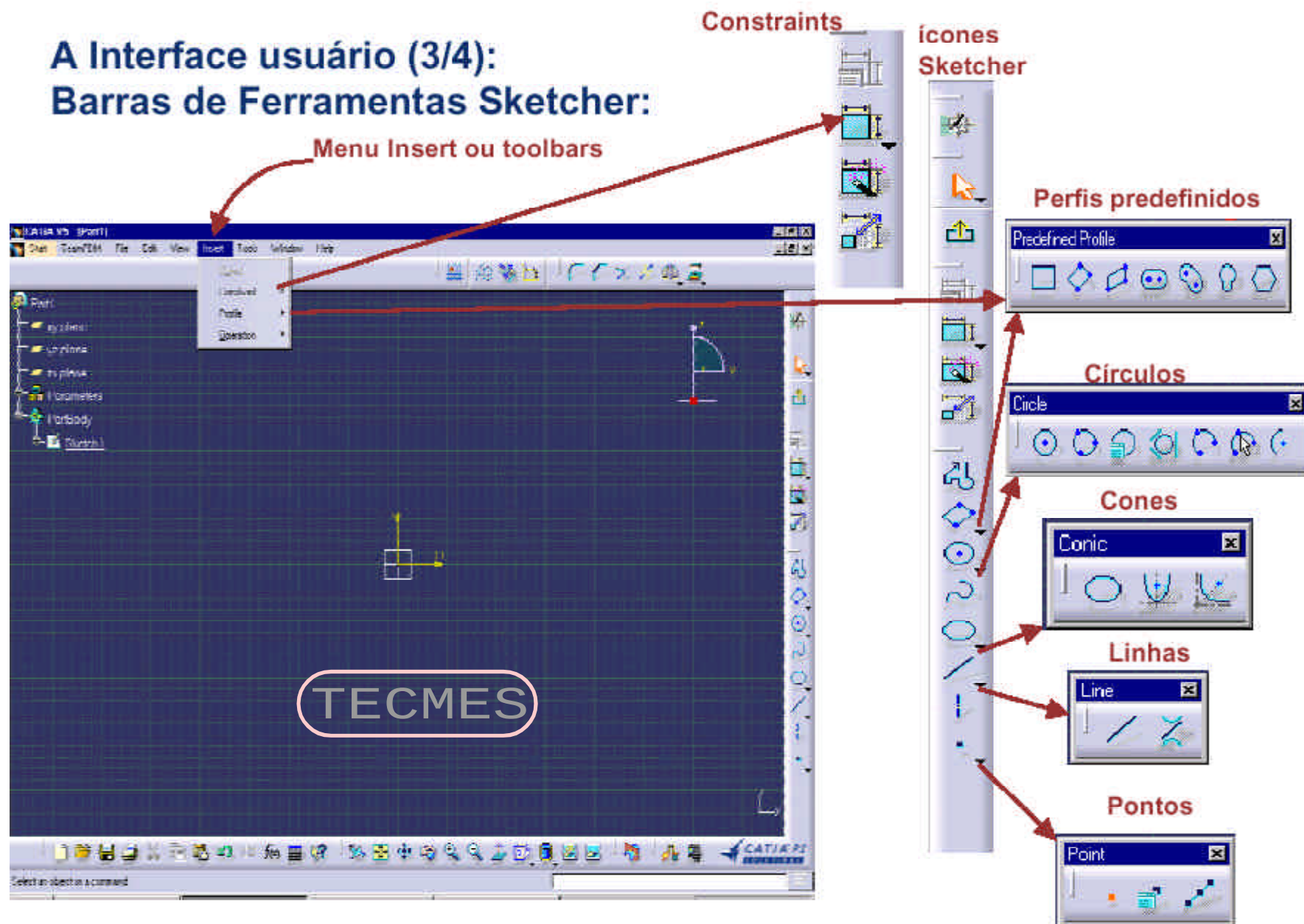
Um novo Sketch vai ser registrado na árvore da peça quando entrarmos no workbench Sketcher



A Interface usuário (2/4): Ferramentas Sketcher



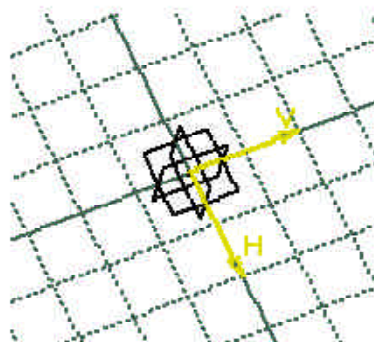
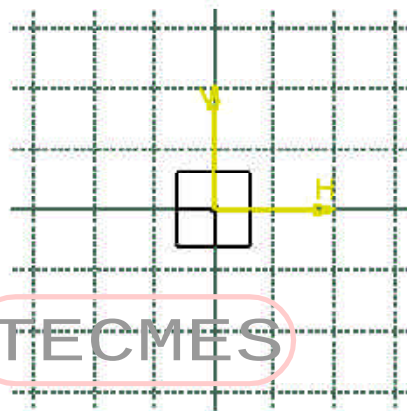
A Interface usuário (3/4): Barras de Ferramentas Sketcher:



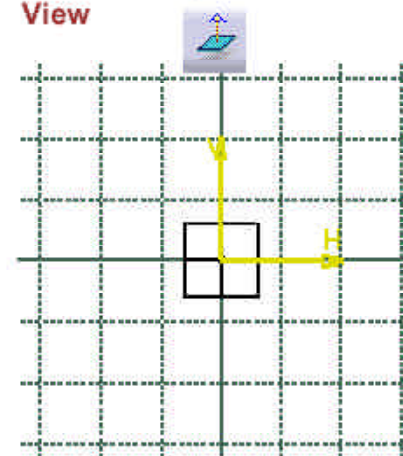
A Interface usuário (4/4): Plano Sketcher:

É possível aplicar zoom, arrastar e rotacionar (usando o mouse)

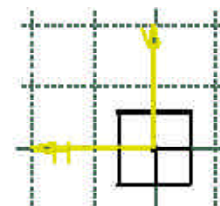
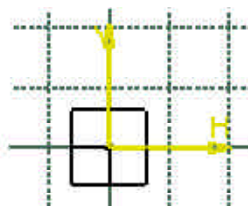
Uma vez que o plano no qual você quer fazer um sketch foi selecionado, ele é mostrado paralelo à tela



Para reiniciar o plano de rotação do sketcher, selecione o ícone Normal View



Se você selecionar o ícone normal view quando o plano do sketch já estiver mostrado paralelo à tela, você retorna o plano sketch para ver seu outro lado.

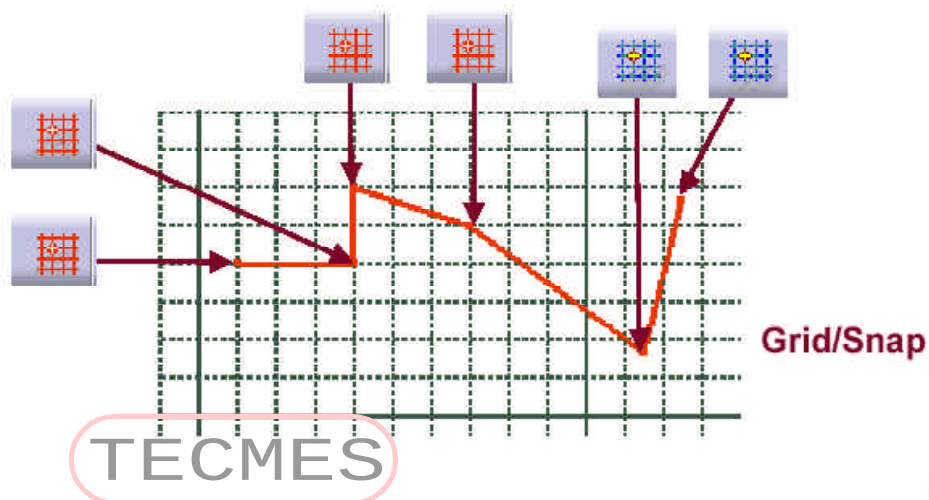


Desenhando Perfis Simples

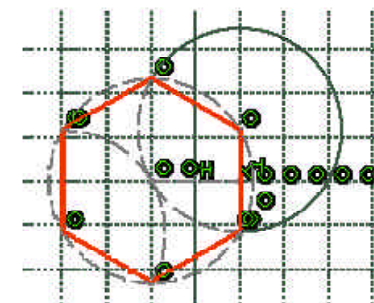
-  **Modo de Trabalho do Sketcher**
-  **Criando Perfis (Profile)**
-  **Criando Pontos**
-  **Criando Linhas**
-  **Criando Splines**
-  **Criando Círculos e Arcos**
-  **Criando Cônicas**
-  **Criando Eixos**
-  **Criando Perfis Pré Definidos**

TECMES

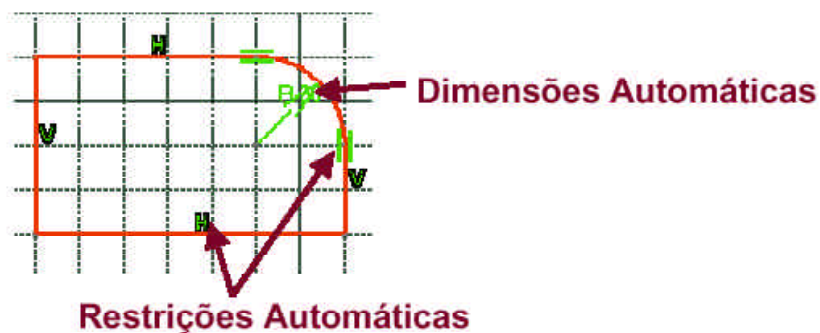
Modo de trabalho do Sketcher



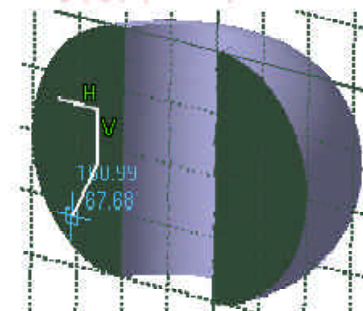
Standard/Construction



Campos de Valores



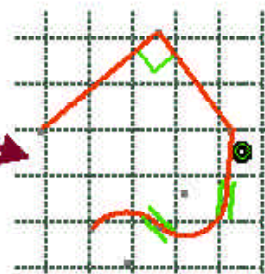
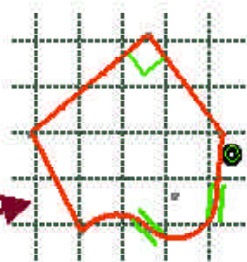
Section View



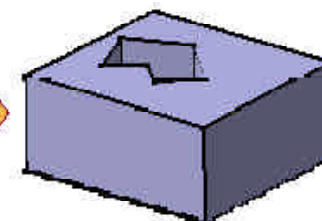
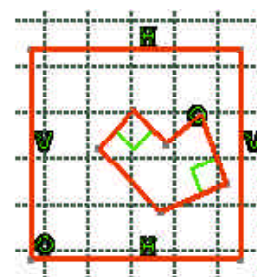
Criando Perfis (Profile)

Perfis podem ser :

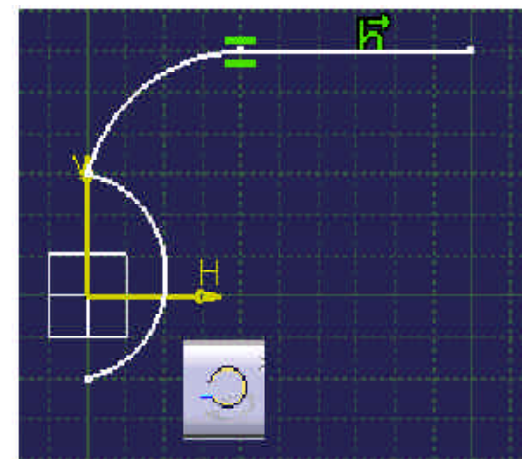
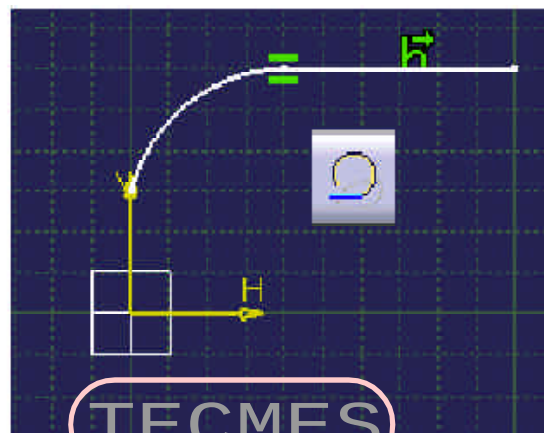
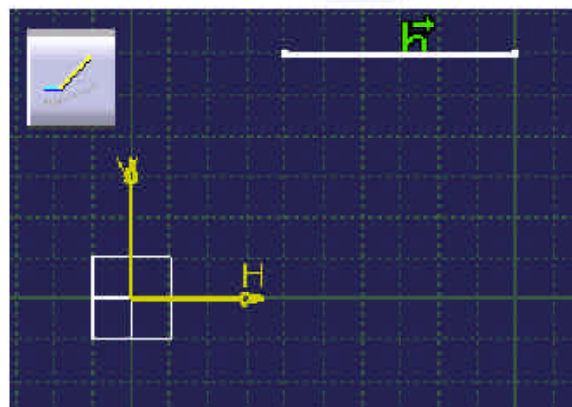
- Fechados
- Abertos



Inner profile

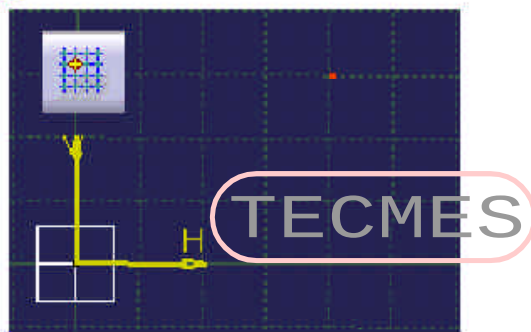


Criação de Perfis

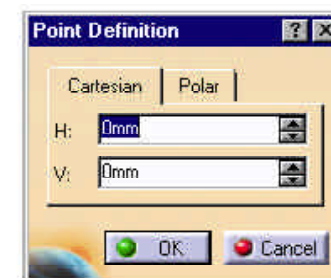


Criando Pontos

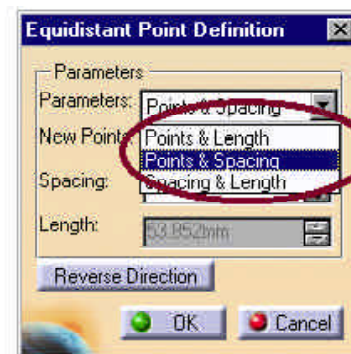
Pontos por click



Pontos por Coordenadas

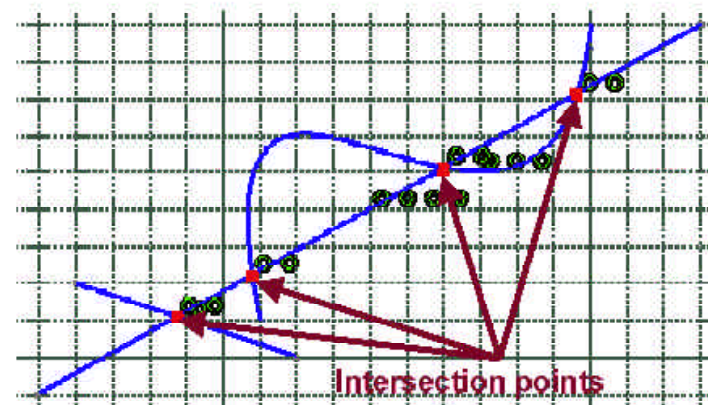
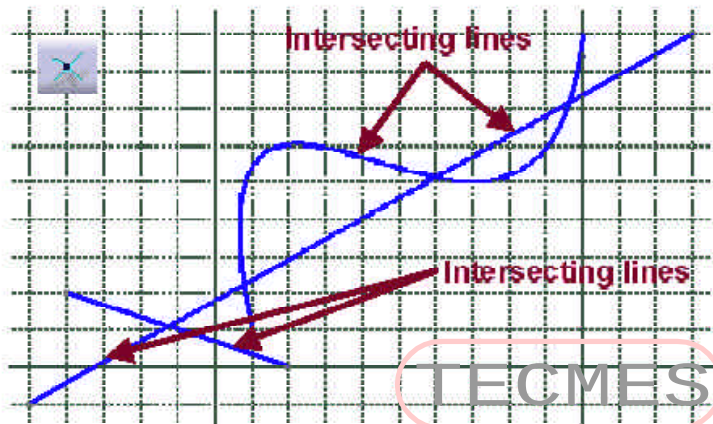


Pontos Equidistantes

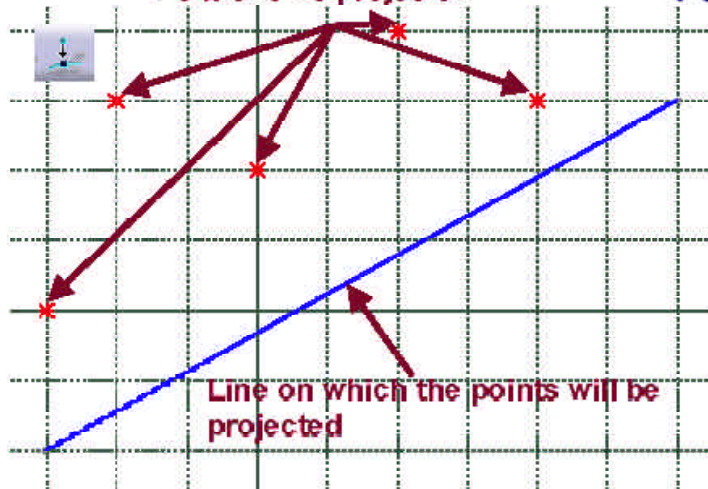


Criando Pontos

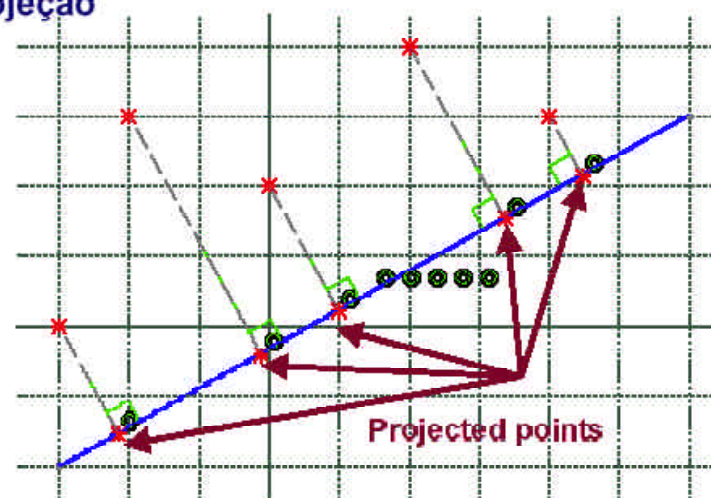
Pontos por Intersecção



Points to be projected



Pontos por Projeção

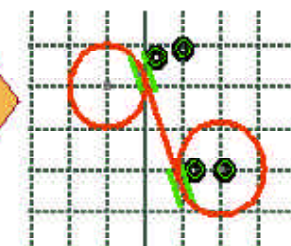
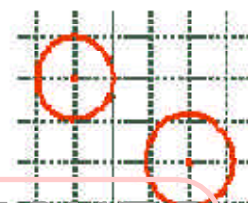


Criando Linhas

Linhas entre dois Pontos

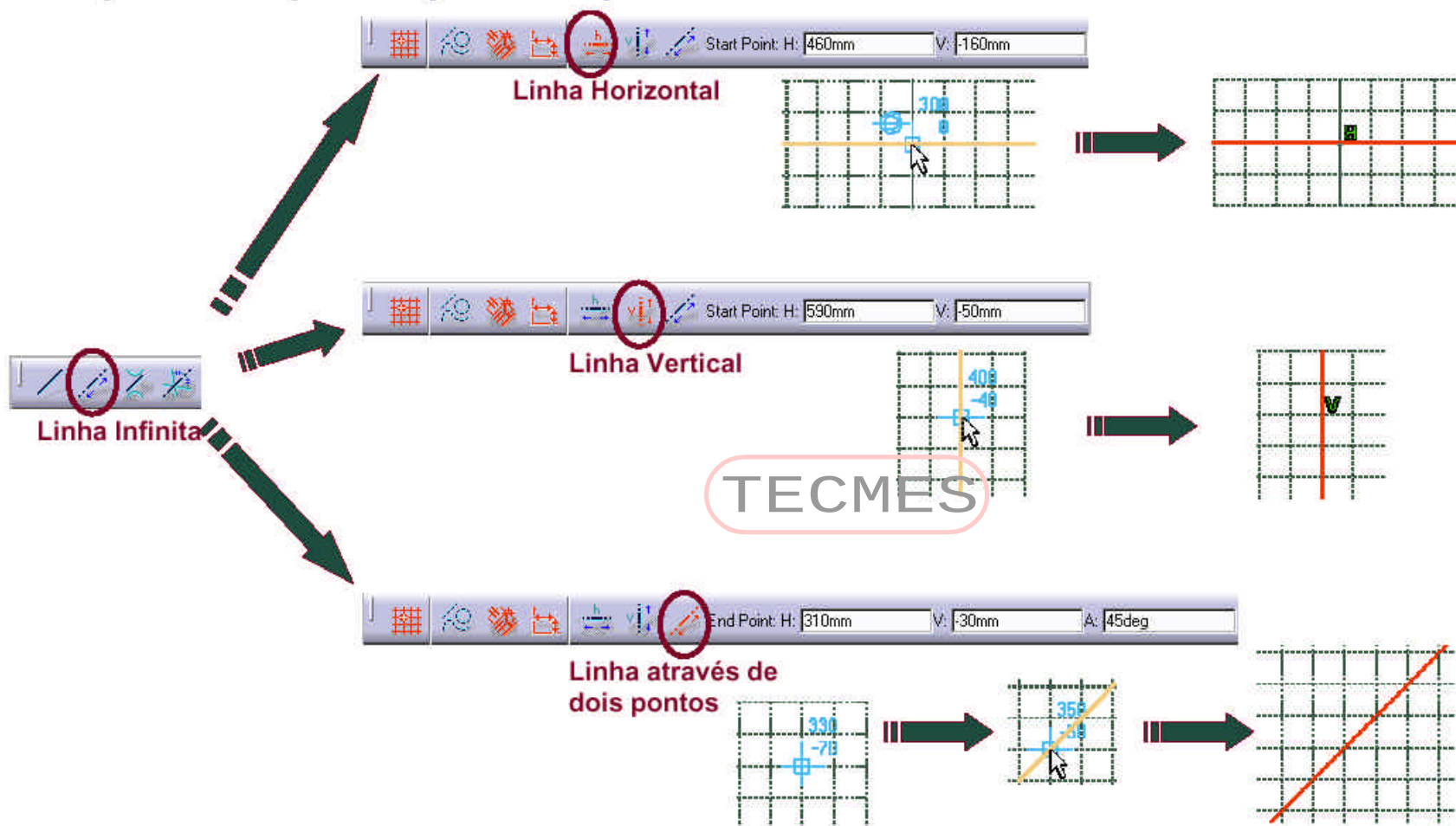


Linhas Bitangentes



Criando Linhas

- O ícone de linha infinita permite a criação de linhas horizontais e verticais passando por um ponto ou por dois.

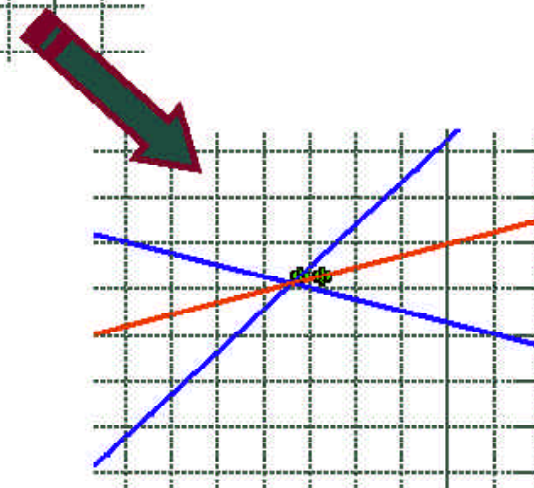
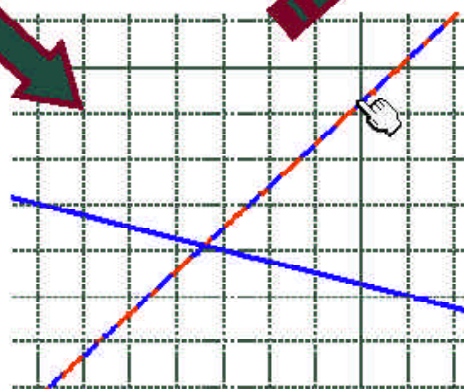
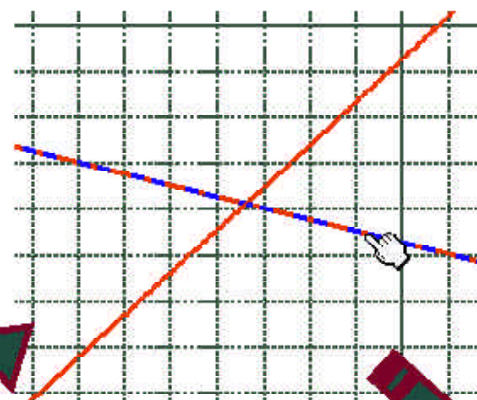
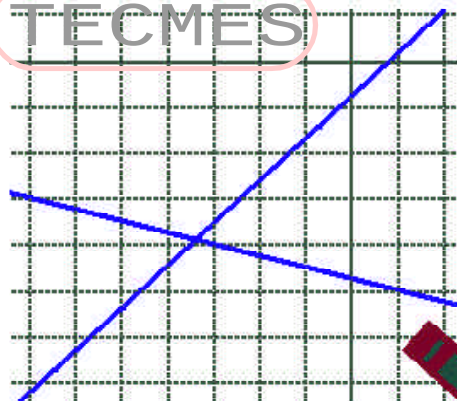


Criando Linhas

● O ícone Bisecting line cria linhas bissetrizes infinitas através de duas outras

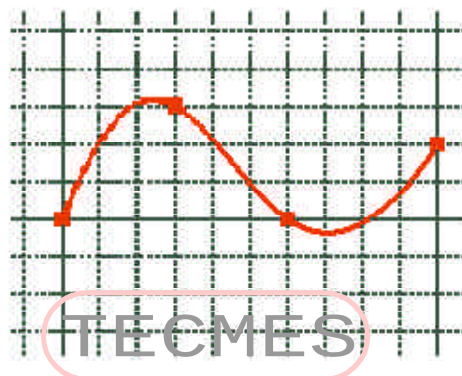


TECMES



Criando Splines

Criando uma SPLINE



Pontos de Controle

Control Point Definition

Cartesian Polar

H: 110mm

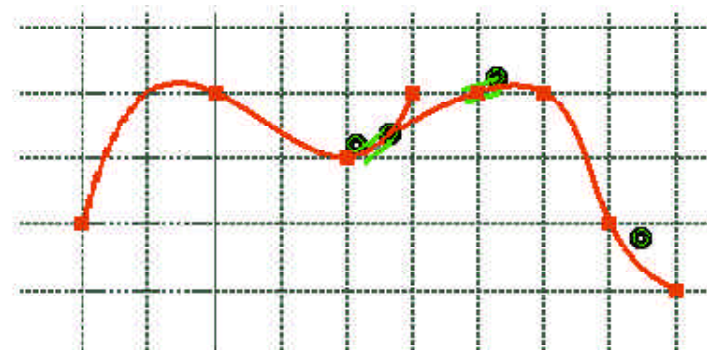
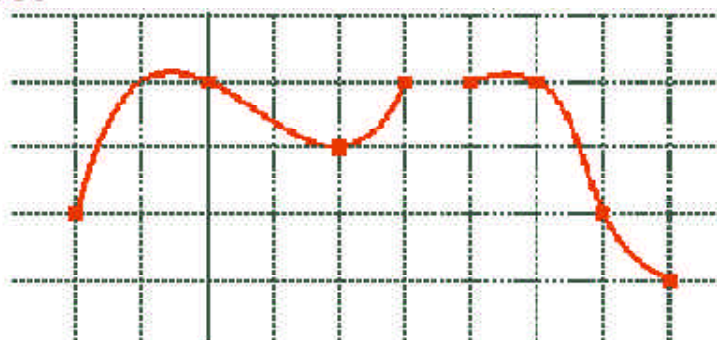
V: -40mm

☐ Tangency Reverse Tangent

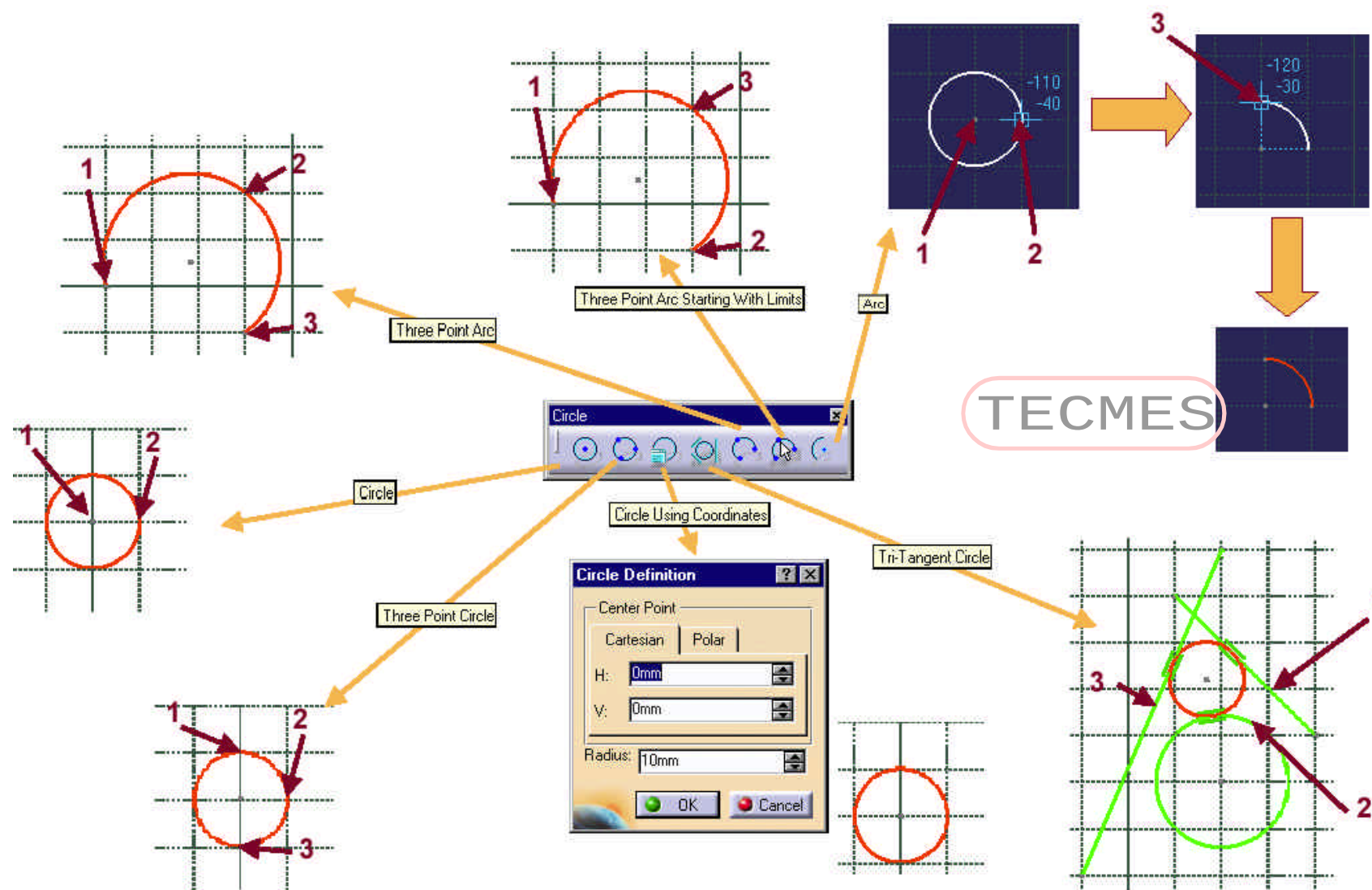
☐ Curvature Radius: 259mm

OK Cancel Help

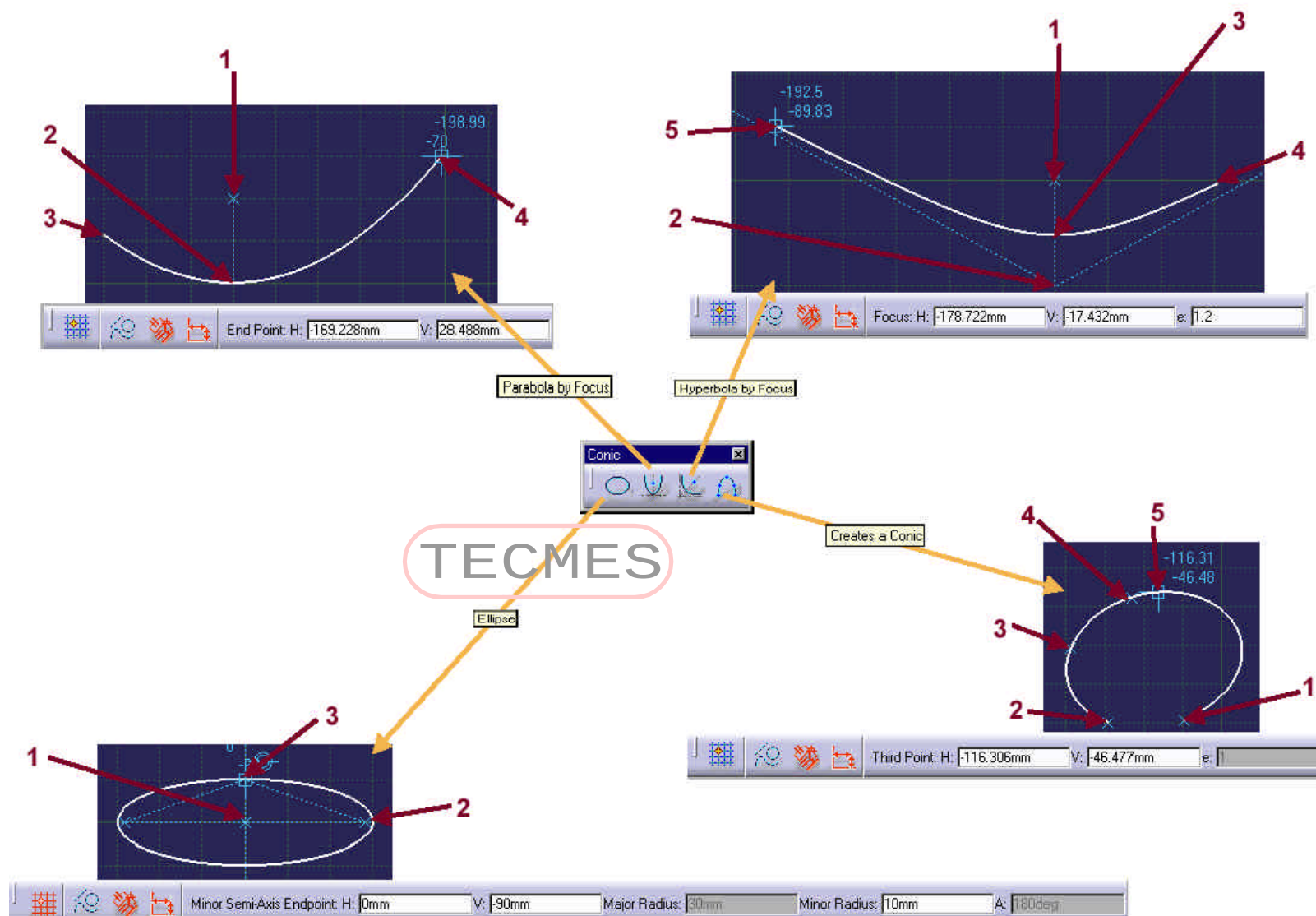
Connect



Criando Círculos e Arcos

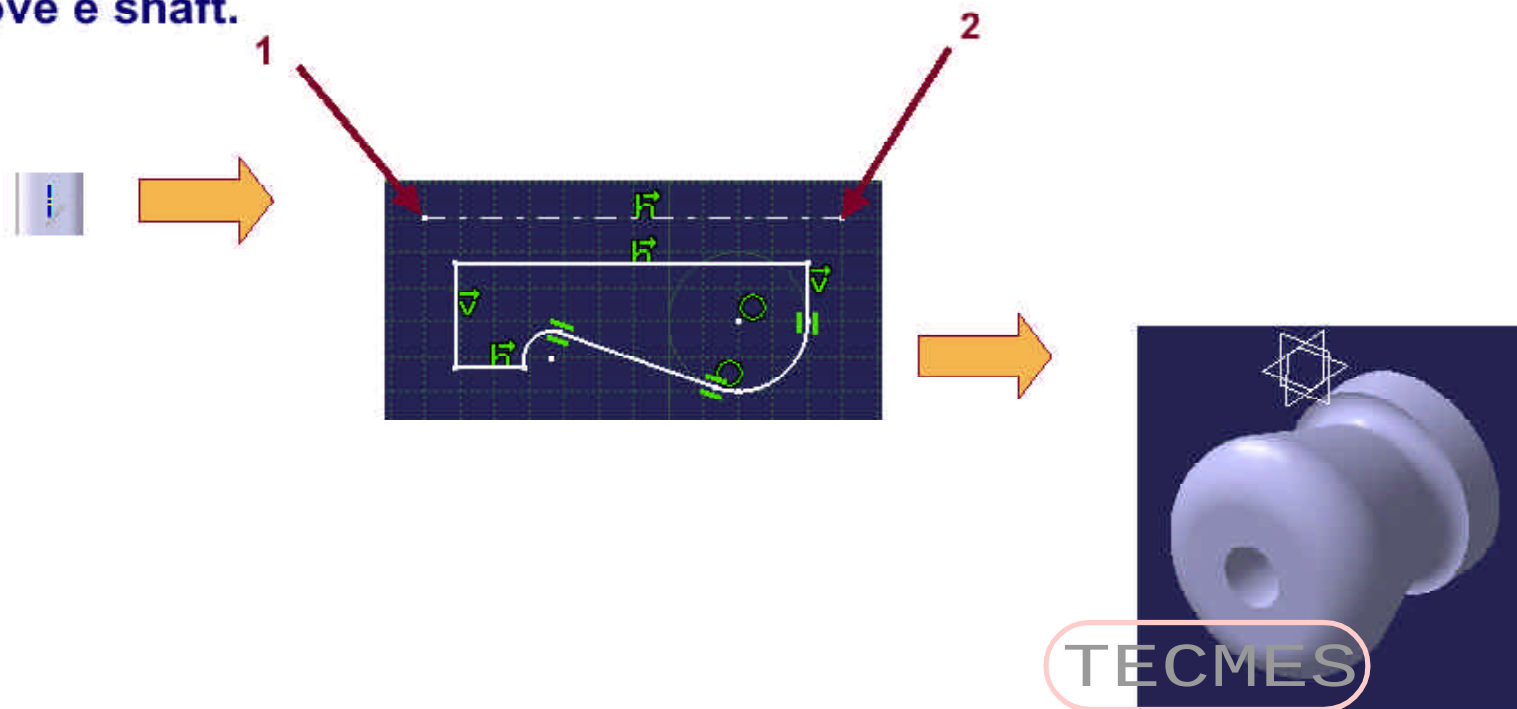


Criando Cônicas



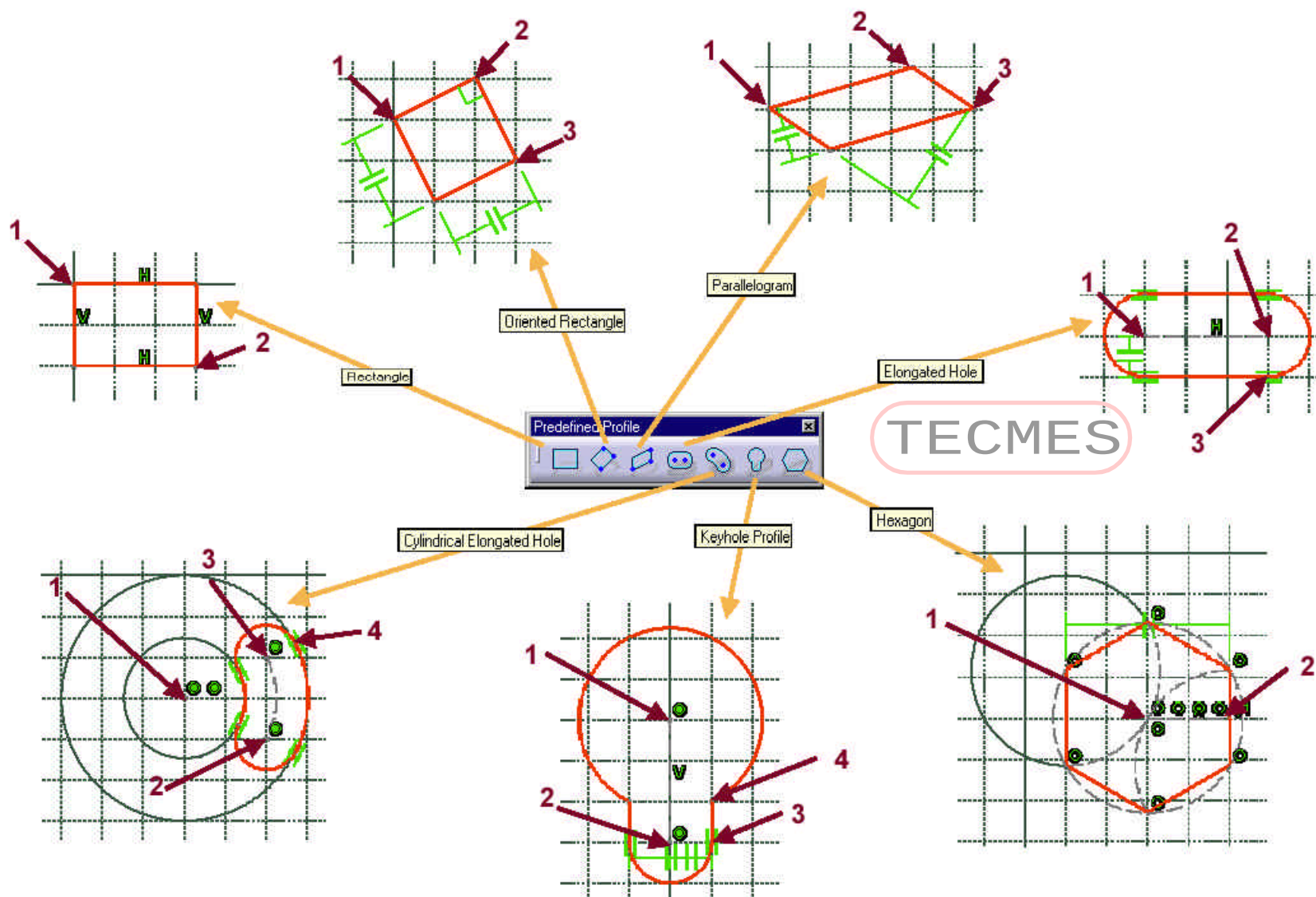
Criando Eixos

- Um eixo será utilizado na criação de uma simetria ou no uso das funções groove e shaft.



TECMES

Criando Perfis Pré-Definidos



Editando Perfis



Modificando Perfis



Fillets e Chanfros



Trim e Break



Quick Trim



Close



Complement



Transformações

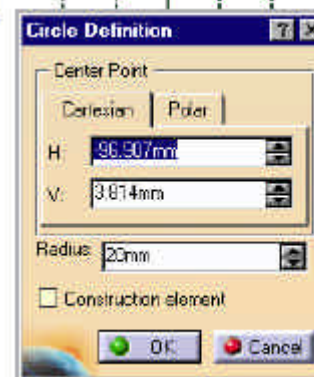
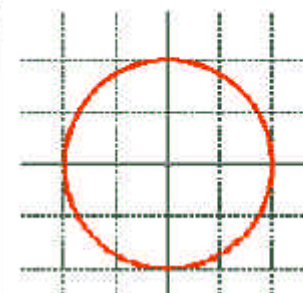
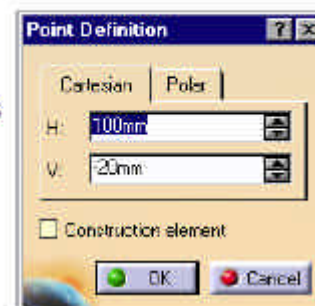
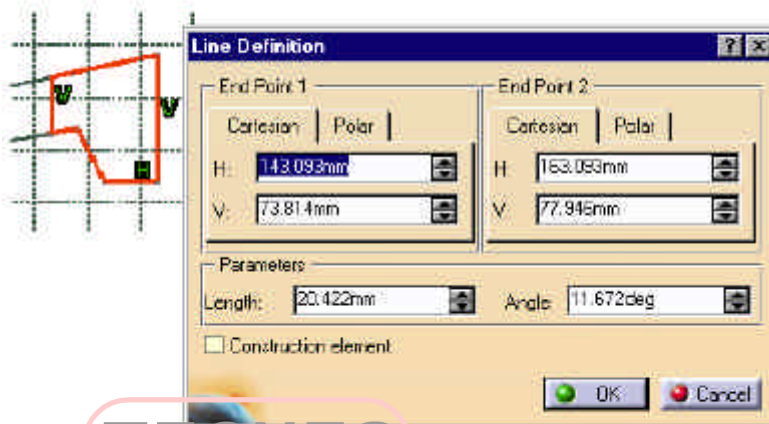


Offset

TECMES

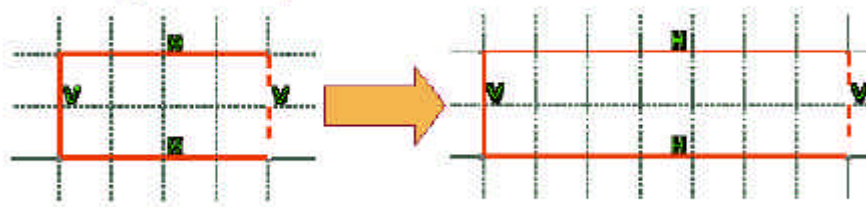
Modificando Perfis

Modificando elementos e propriedades dos perfis

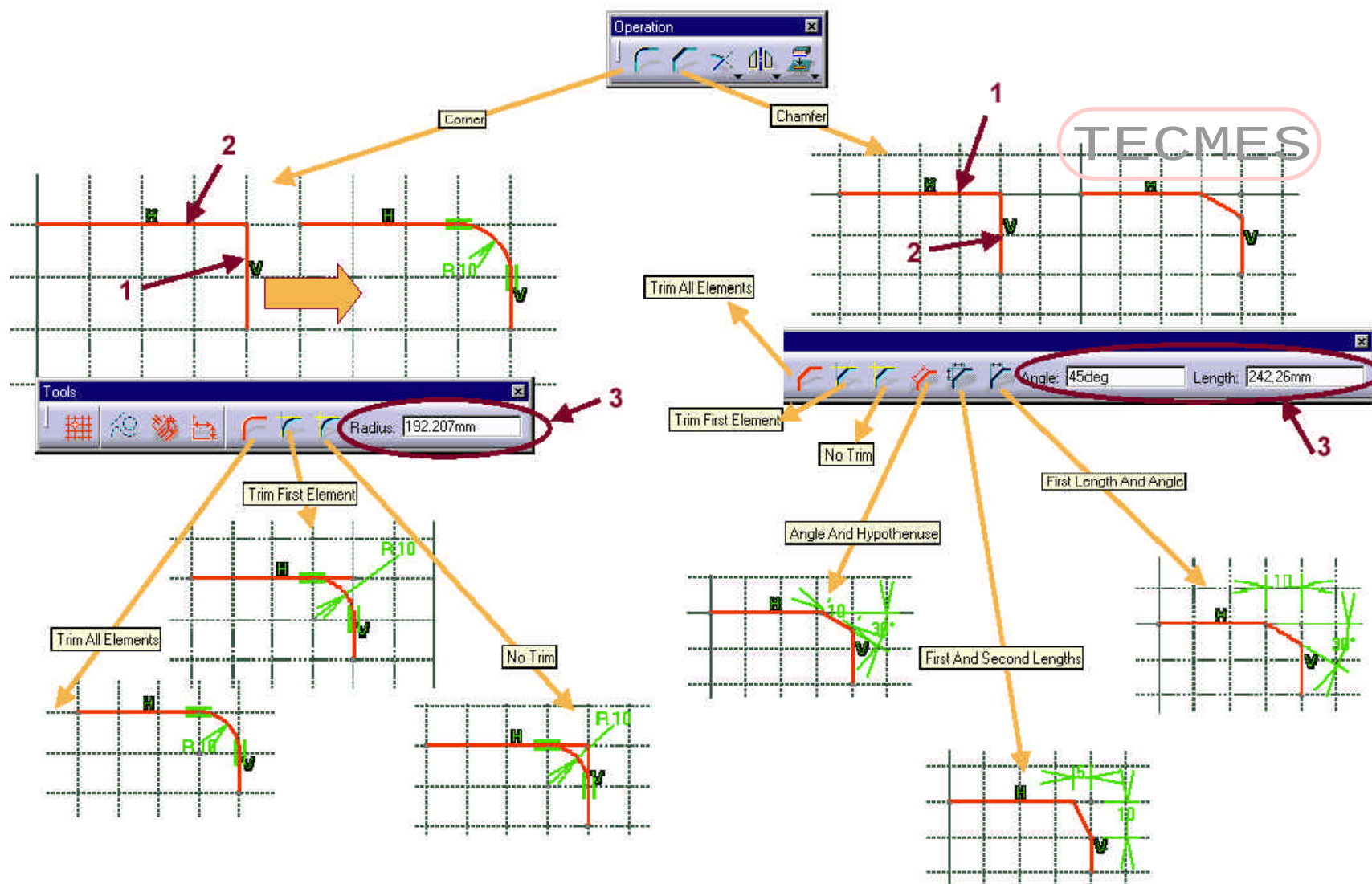


TECMES

Drag and drop elements

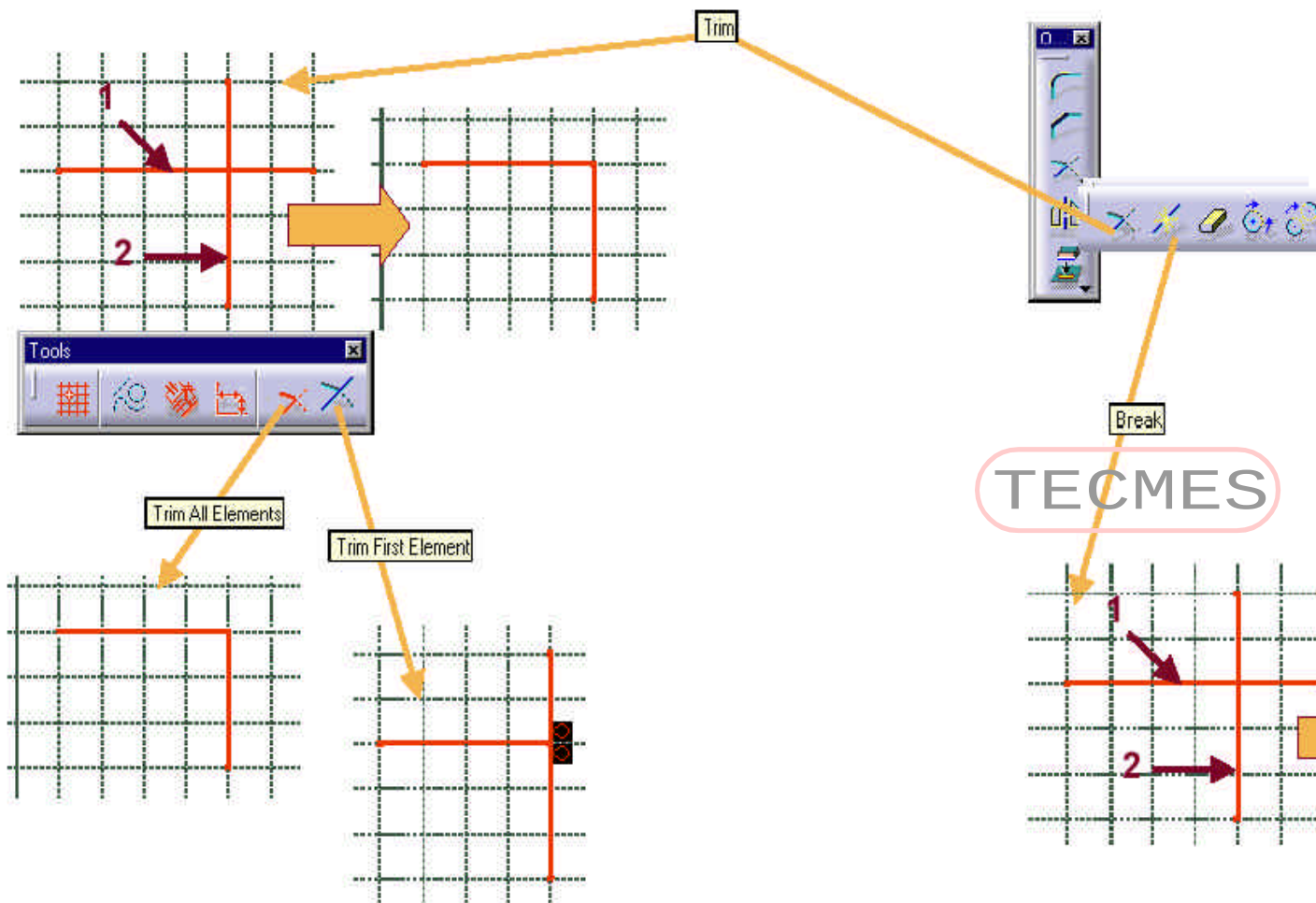


Fillets e Chanfros



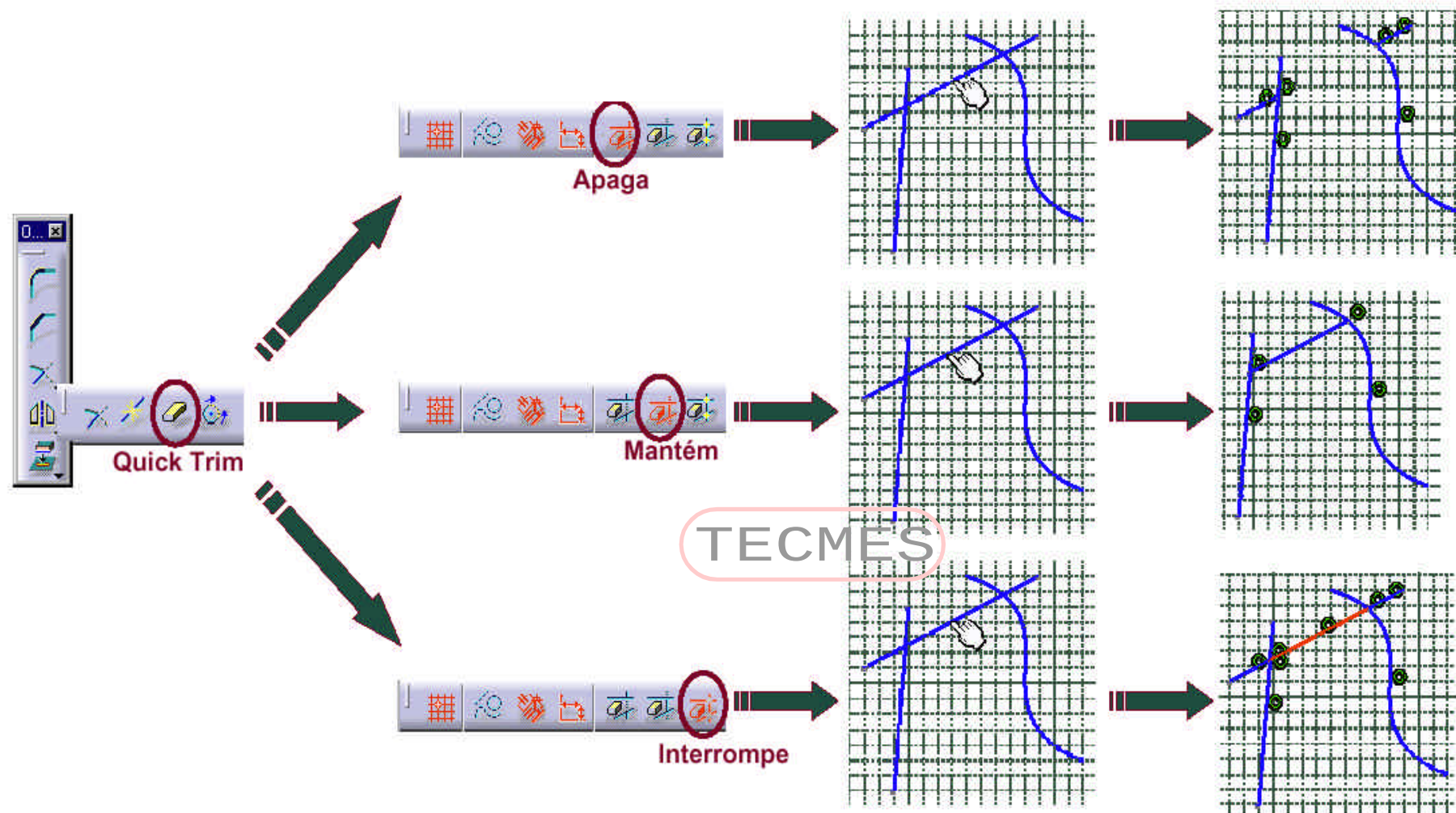
Esta apostila é parte integrante do material de treinamento TECMES, não podendo ser copiada ou reproduzida integral ou parcialmente sem prévio consentimento.

Trim e Break

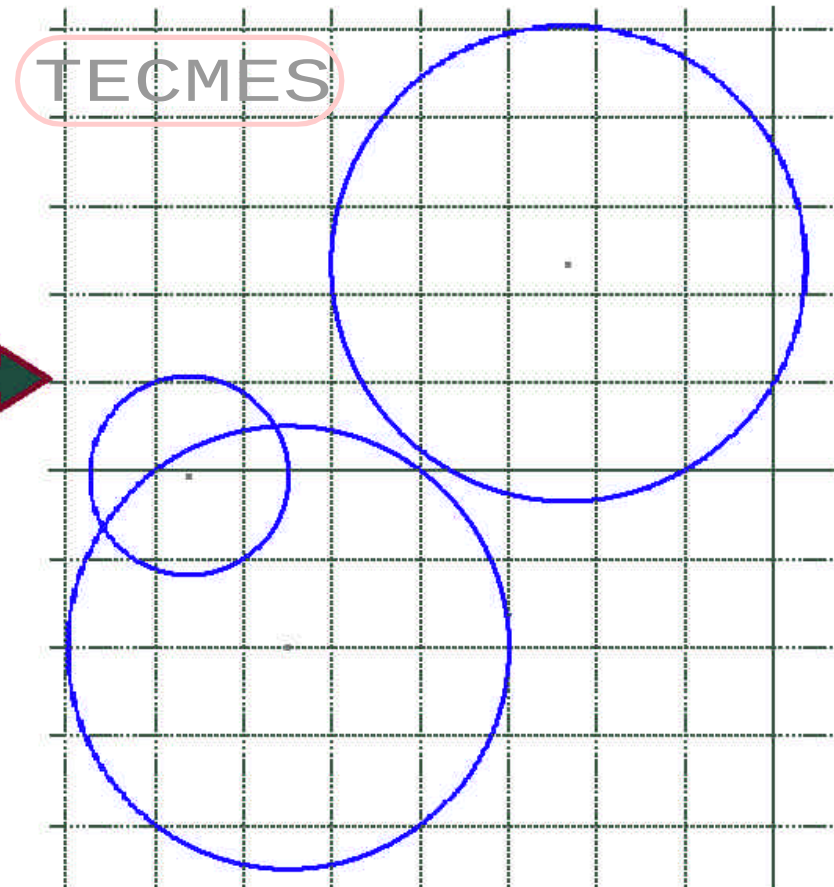
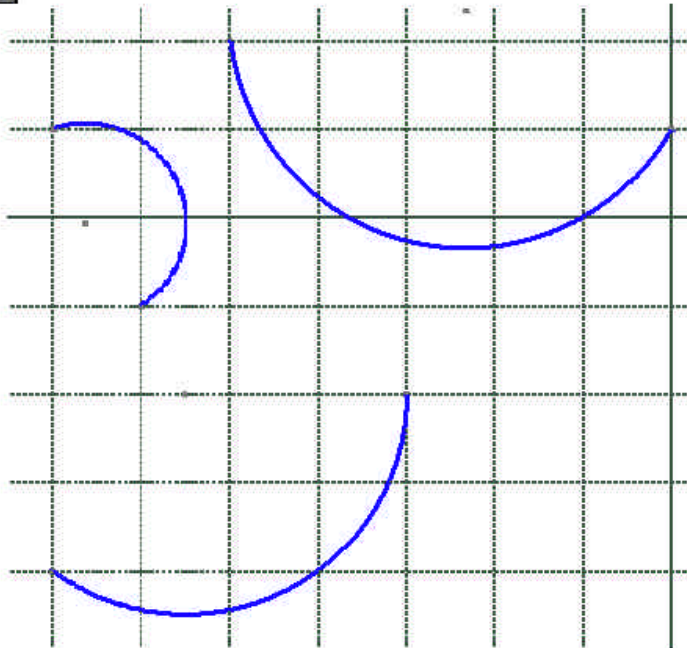


Esta apostila é parte integrante do material de treinamento TECMES, não podendo ser copiada ou reproduzida integral ou parcialmente sem prévio consentimento.

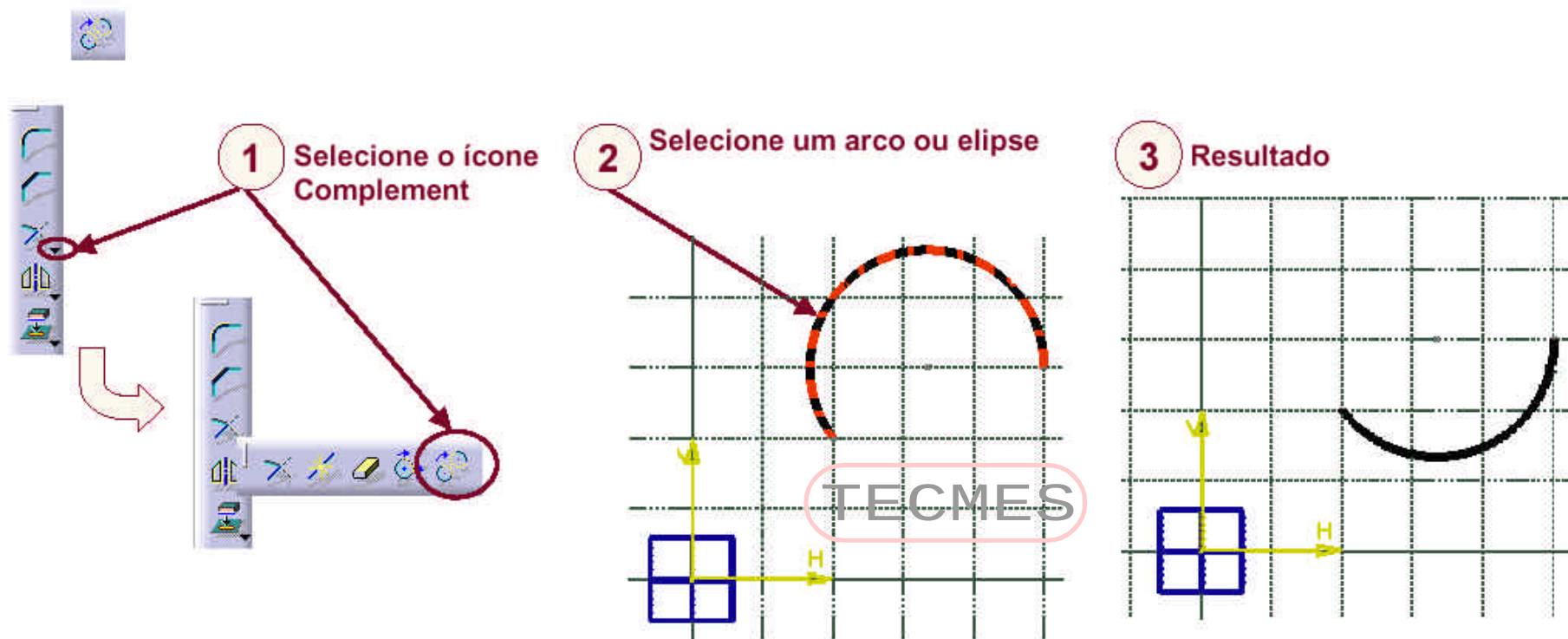
Quick Trim



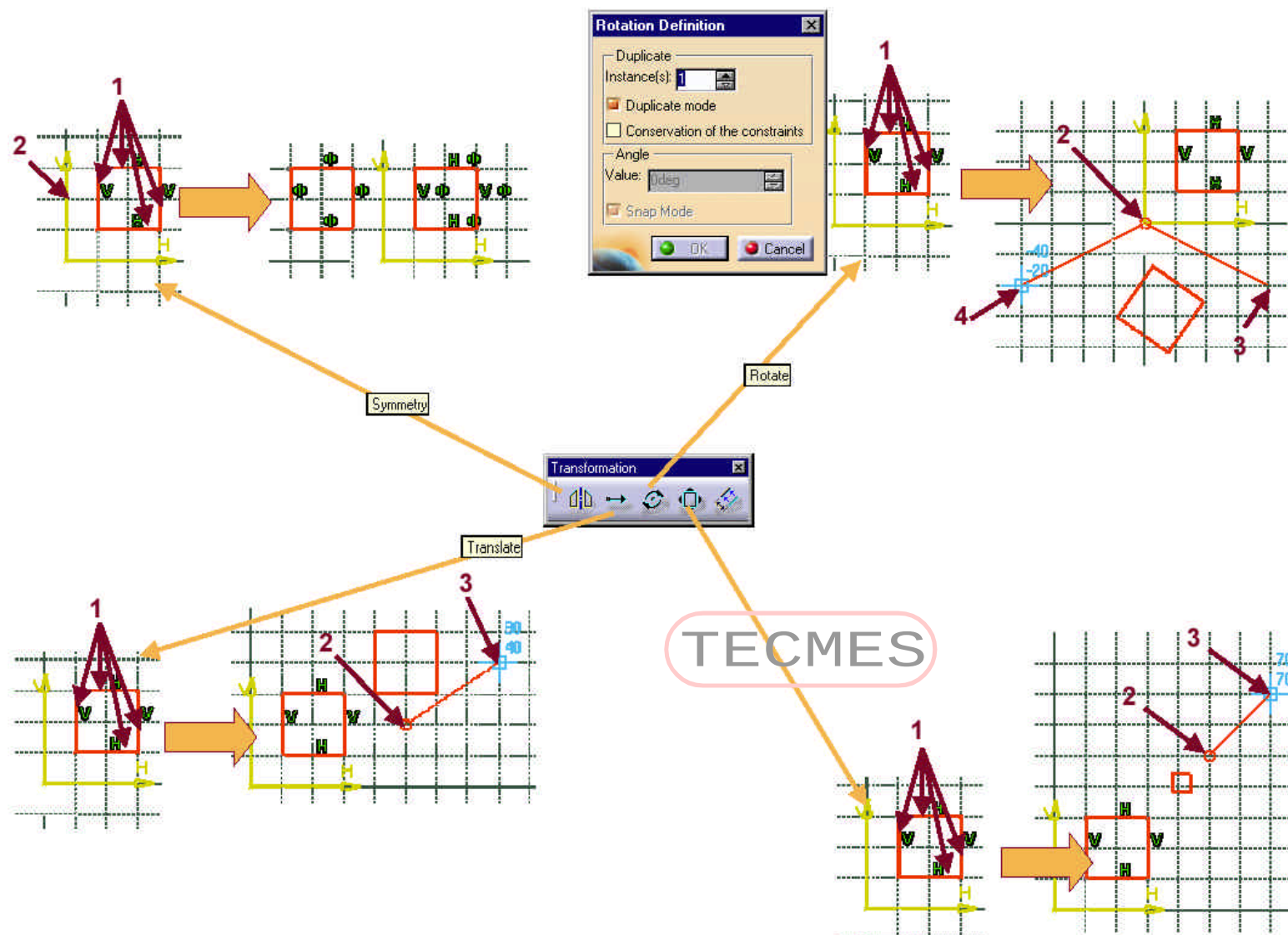
Close



Complement

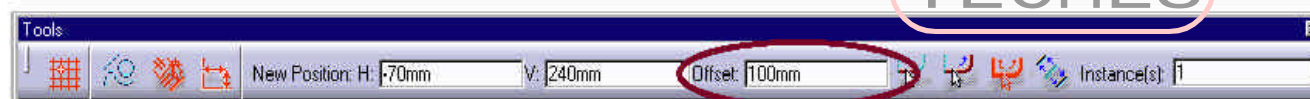
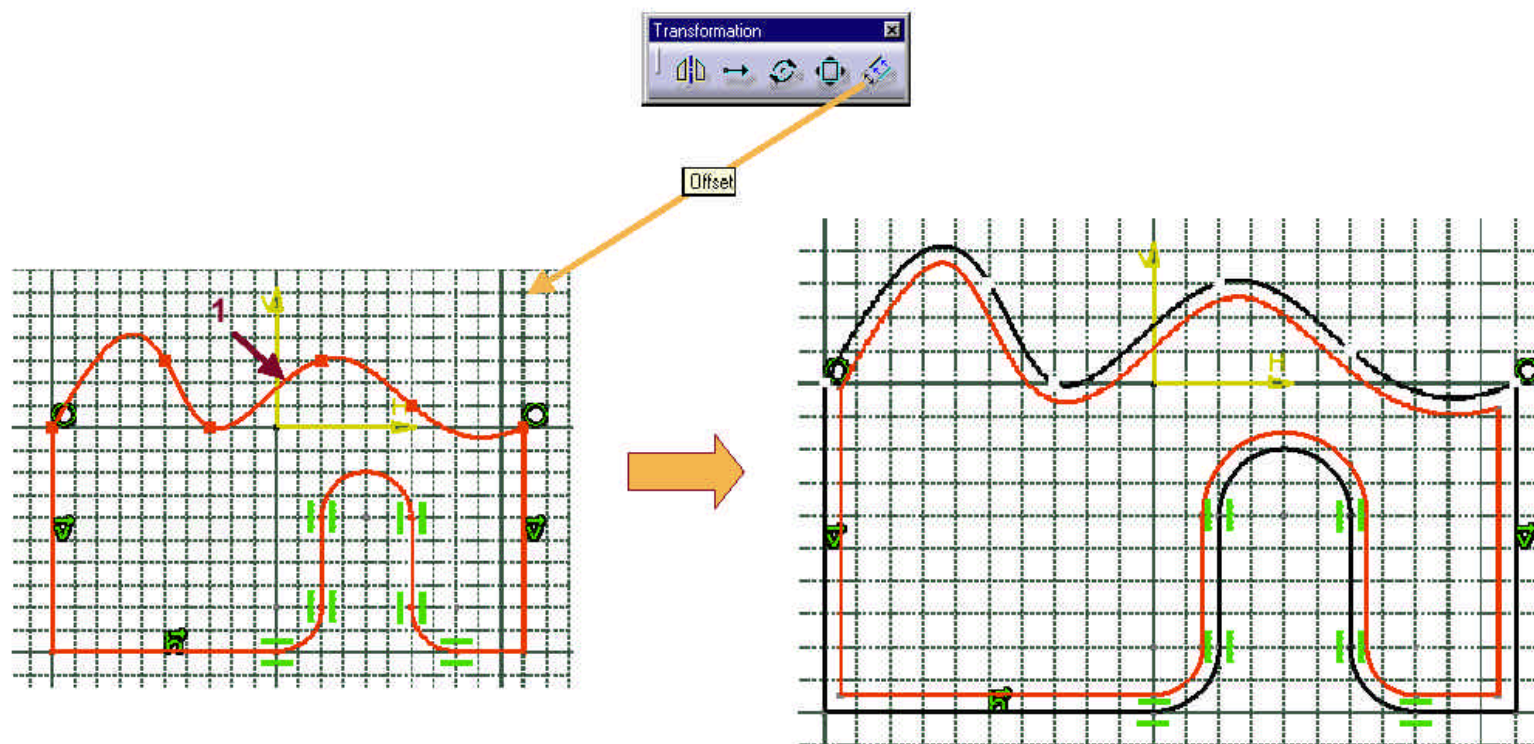


Transformações



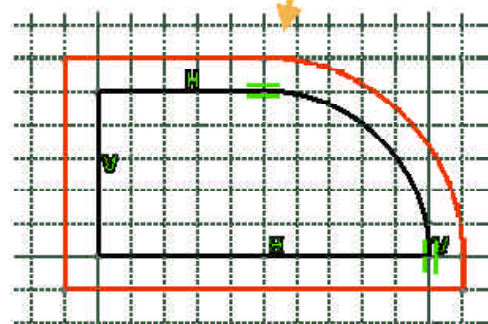
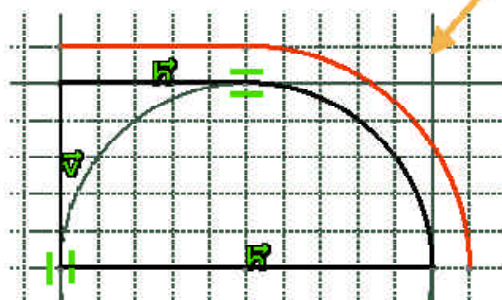
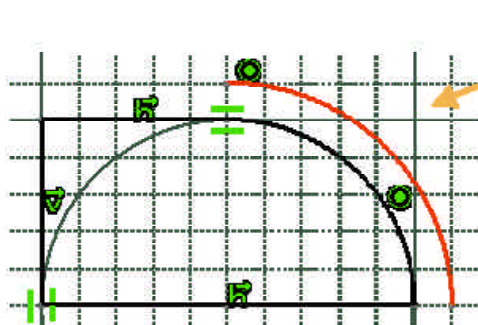
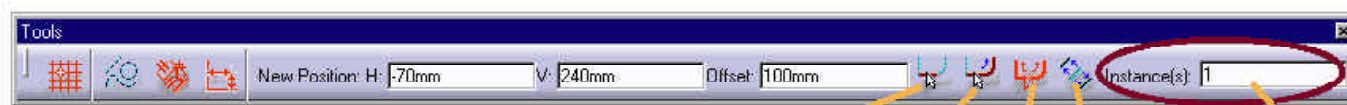
TECMES

Offset

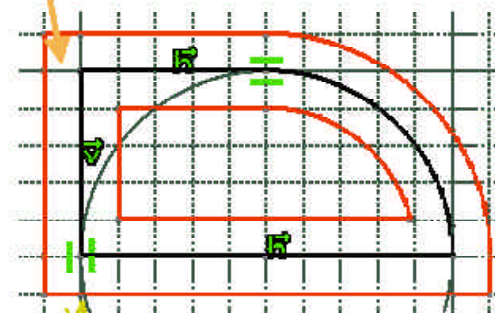
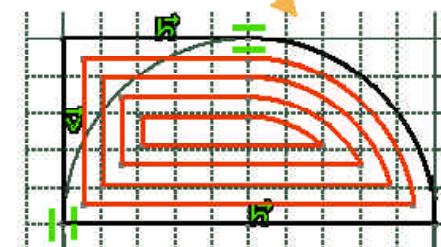


2

Offset



TECMES



Restrições (Constraints)



Criando Restrições (Constraints)



Modificando Restrições (Constraints)



Criando Restrições Automaticamente
(Auto Constraint)



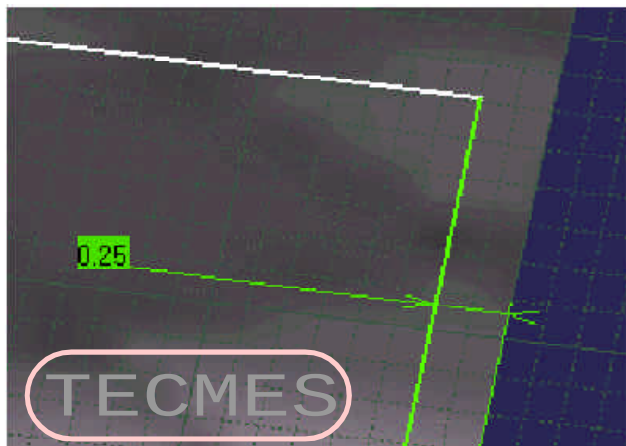
Animando Restrições (Constraints)



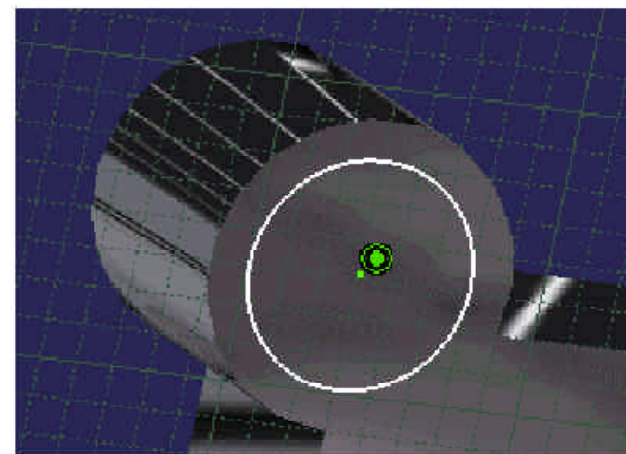
Relacionando Restrições (Constraints)

TECMES

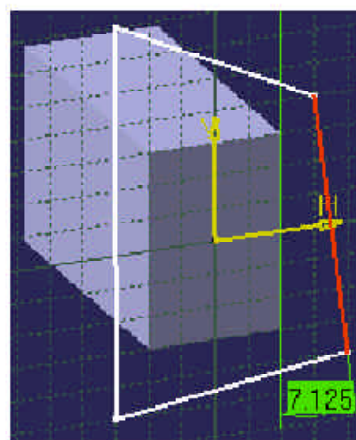
Criando Restrições (Constraints)



**Restrições de Dimensão
(Constraints)**

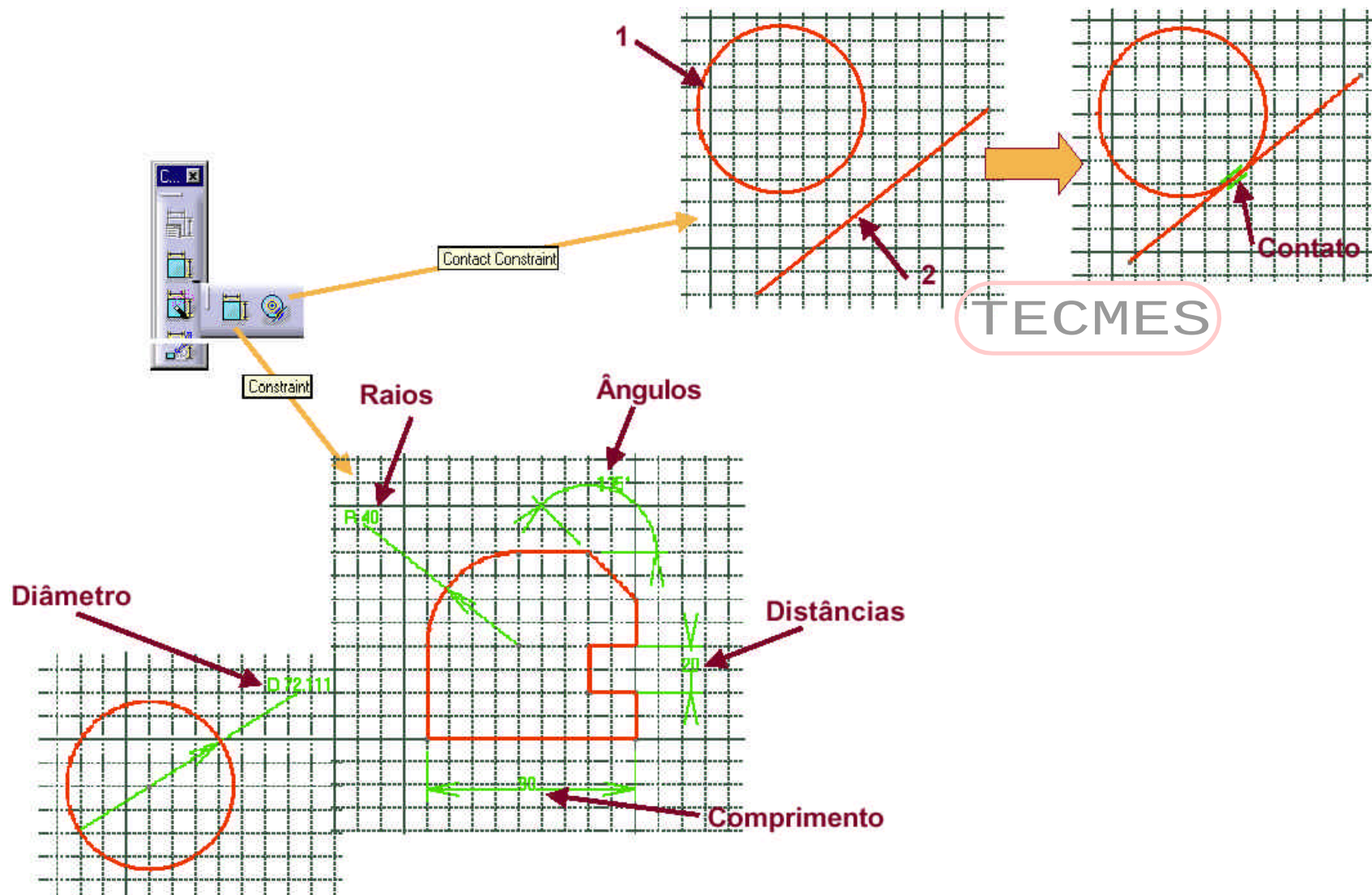


Restrições Geométricas

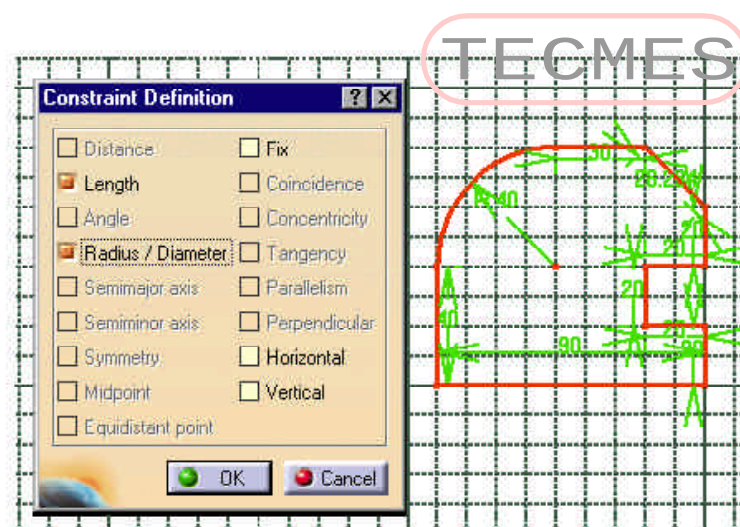
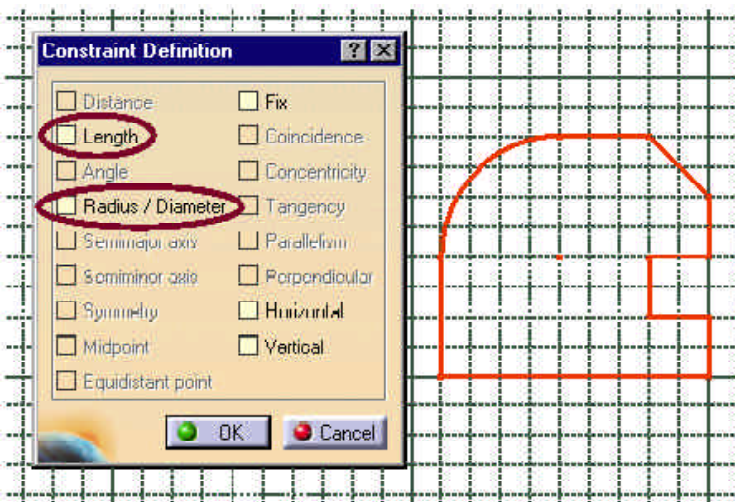


Criando perfis "In Context"

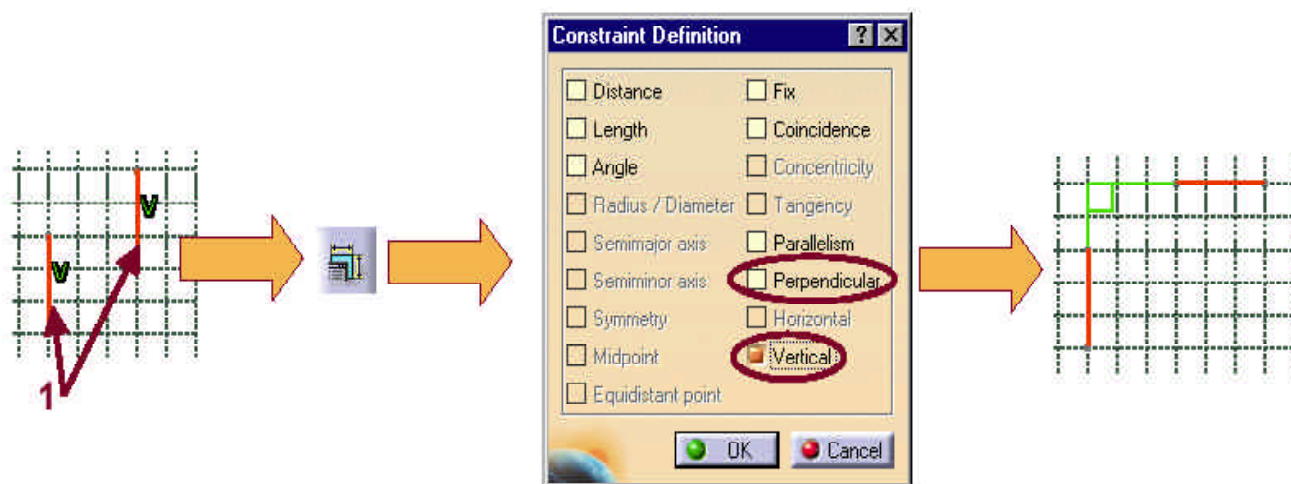
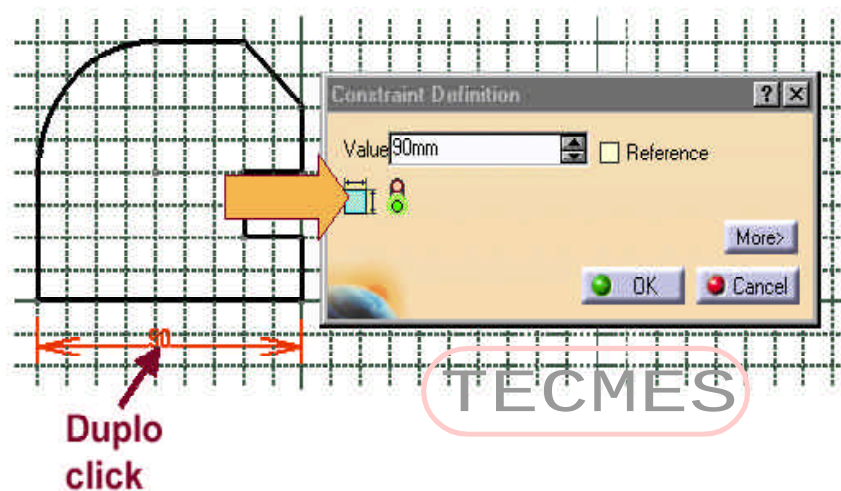
Criando Restrições (Constraints)



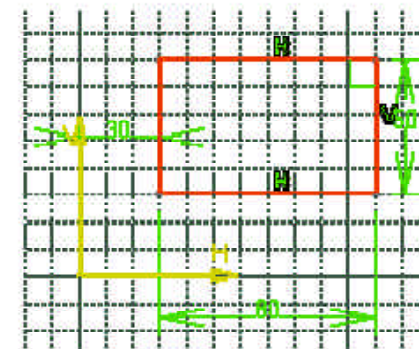
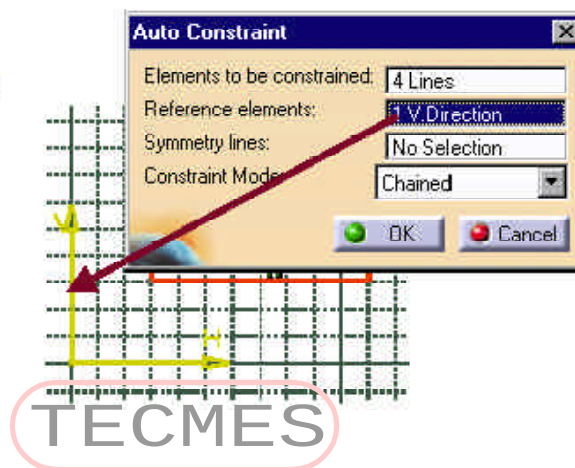
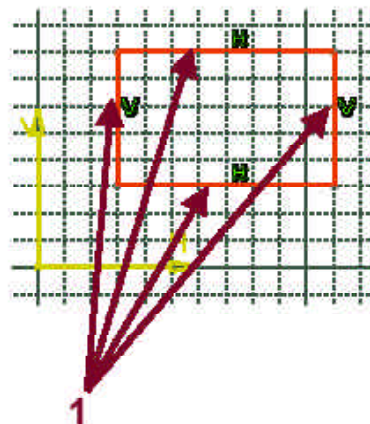
Criando Restrições (Constraints)



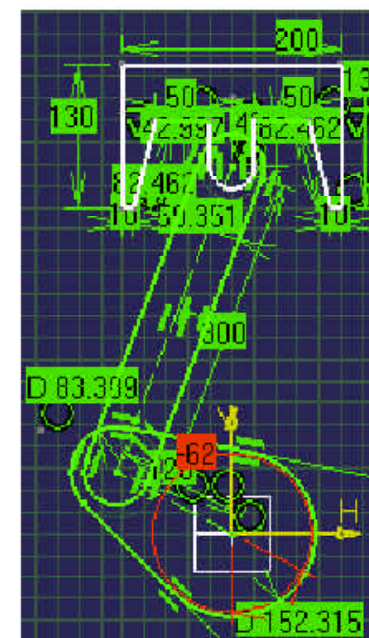
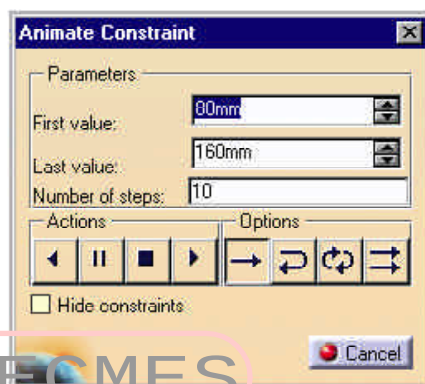
Modificando Restrições (Constraints)



Criando Restrições Automaticamente (Auto Constraint)



Animando Restrições (Constraints)



Relacionando Restrições (Constraints)

The diagram illustrates the process of relating constraints in CATIA V5 Sketcher. It shows three sketches connected by arrows, representing a sequence of steps:

- Step 1:** A sketch on a grid with a vertical dimension of 40 and a horizontal dimension of 80. A red arrow points to the horizontal dimension line.
- Step 2:** A context menu is shown for the 'Length.84 object'. The 'Edit formula' option is selected. This leads to the 'Formula Editor' dialog box, which displays the formula: $\text{PartBody\Sketch.1\Length.13\Length} + 20$. The 'Dictionary' tab is active, showing 'Parameters' and 'Length'.
- Step 3:** The final sketch shows the horizontal dimension updated to 60, indicating the result of the formula application.

