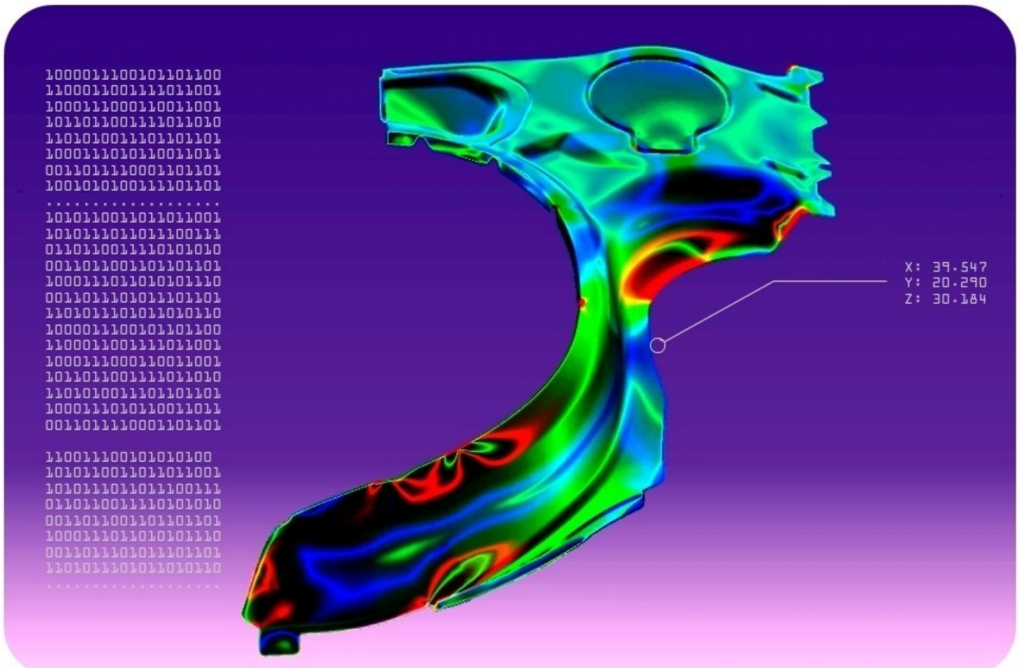
FTICATIA V5

User Guide



CAA V5 BASED

HEXAGON

HEXAGON

Hexagon is a global leader in sensor, software and autonomous solutions. We are putting data to work to boost efficiency, productivity, and quality across industrial, manufacturing, infrastructure, safety, and mobility applications.

Our technologies are shaping urban and production ecosystems to become increasingly connected and autonomous – ensuring a scalable, sustainable future.

Learn more at Hexagonmi.com. Hexagon's Manufacturing Intelligence division provides solutions that utilise data from design and engineering, production and metrology to make manufacturing smarter.

Learn more about Hexagon (Nasdaq Stockholm: HEXA B) at hexagon.com and follow us @HexagonAB.

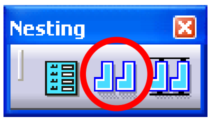
Disclaimer

Contents

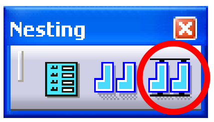
Chapter 1: Introduction

FTICATIA is cost optimization, nesting, and formability simulation software integrated within the CATIAV5 environment. The main modules include CATNEST, PROGNEST, CATBLANK, CATFORM, CATSTAMP, COST OPTMIZER and COST OPTIMIZER ADVANCED.

CATNEST: Specifically designed for nesting sheet metal blanks on coils.



PROGNEST: Specifically designed for producing the initial strip layout on coils for progressive die forming operations.



CATBLANK: Specifically designed for blank development and cost estimating.

- For process conditions, only CLAMPS, User Defined Restraint and Enforced Displacement are available.
- Curved Binder analysis is available, but Tailor Welded Blank analysis is NOT available
- For formability result, only THICKNESS is available.

CATFORM: Specifically designed for cost estimating and formability analysis.

- For process conditions, Draw Bead is NOT available.
- Tailor Welded Blank analysis is available, but Curved Binder analysis is NOT available.
- For formability result, limited forming results are available.

CATSTAMP: Specifically designed for cost estimating and complete formability analysis with complete forming conditions and result plots available.

COST OPTIMIZER: A combination of CATBLANK, CATNEST and Cost Optimization.

COST OPTIMIZER Advanced: A combination of CATSTAMP, CATNEST and Cost Optimization.

Chapter 2: Preparing the Model for Analysis

In this section:

- Preparing the Surface Model
- Hole Filling & Healing
- Features Added in Secondary Operations
- Defining the Material and External View

Preparing the Surface Model

The mesher that is used in the FTI analysis tools is designed to mesh native CATIA data. When the geometry is clean (as native geometry usually is), there are almost never any problems with mesh generation. When trying to mesh imported data however, there are sometimes problems. These issues are almost universally caused by problems with the geometry. These problems include poor surface boundaries, twisted surfaces and overlapping surface edges.

Preparing Imported Surface Geometry

When surface data is imported in IGES format, the individual surfaces are not connected. In order to mesh the part, the surfaces must be joined together to form one unit – a continuous "quilt" of surfaces.

There are two functions in CATIAV5 that can help you join the surfaces. The first function is called Join. This tool will use the imported surfaces without modification and simply recognizes where they should be joined. Alternatively, the Heal function actually modifies the surfaces in an attempt to close gaps. While this function will accept wider tolerances, it sometimes creates new problems while fixing others.

Preparing Surface Geometry from Solid Model

If you are working with a solid model, a surface model can be created by using the Extract function from the Operations toolbar in the Generative Shape Design workbench.

You can use Top, Bottom, or Middle surface from the solid model for analysis. You simply must ensure that you choose the appropriate thickness direction during analysis setup.

Hole Filling and Healing

If holes in the part are pierced in a secondary operation after the forming operation, the holes should be removed from the geometry.

Holes can be filled using CATIA commands such as the **Fill** command. However for parts that contain several holes this can be a time consuming process. The Fill Holes & Heal command will help automate the hole filling process.

The **Fill Holes & Heal** command will automatically fill in all holes that are smaller than a user defined value in the selected geometry with a CATIA Fill surface. The user has the option to choose **Fill or Heal**.

The **Heal** function is designed to help clean up imported or V4 data. It uses specialized algorithms to automatically eliminate duplicate surfaces, repair bad surfaces and correct topology issues. The healing function may modify the base surfaces so it should only be used for preparing a model for analysis. You can perform healing on a single surface body or on multiple surface entities.

The **Fill Holes & Heal** command is found in the **FTI Forming Tools** workbench. Select **Start > Mechanical Design > FTI Forming Tools**. It is also a toolbar that can be added to the Generative Shape Design (GSD) workbench.

Some holes cannot be filled with a single surface. This often occurs when the boundary is too complex (such as on very large holes) or discontinuous. This aspect of the Hole Filling function actually proves very useful in identifying issues with the CATIA geometry since it will show you which holes could not be filled and therefore may contain discontinuous geometry.

If any of the Fill features fail, a warning will be shown in the specifications tree and the feature will not be created. If you expand the Fill object in the specifications tree, it will contain a Boundary object. You can select the Boundary object and then **select Reframe On** in the contextual menu to find this boundary.

Features Added in Secondary Operations

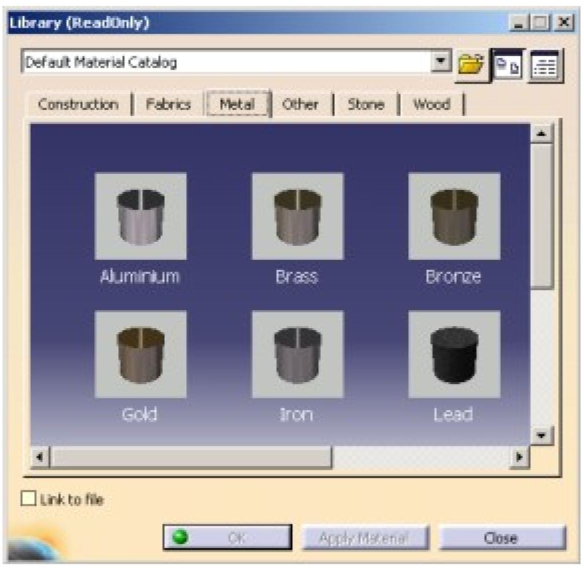
Any features added in secondary operations should be removed using CATIAV5 base functionality. These features include, but are not limited to, holes, notches, extrusions, and hems.

Defining the Material and External View

Although FTICATIA products do not utilize the properties of the material assigned to the part model, all of CATIA's analysis tools require that a material be assigned to the model prior to beginning an analysis. This requirement translates to FTICATIA products as well.

To assign a material to your model, select the **Apply Material** icon from the Apply Material toolbar in the **Generative Shape Design** workbench.

The dialog shown at left will appear. Select a material and select the geometry to which you want to apply the material. The easiest way to do this is to select the **Part1** item at the top of the feature tree. This will apply the material to the entire model.



If you apply a material that has the same name as a material in the FTI material database, then you will not have to apply the material again in the analysis. We have provided another material catalog with all of the materials in our database. It can be found in C:\FTI\FTICATIAV5Rxx and is named FTICatalog.CATMaterial.

The final step in preparing your model for analysis is defining the **External View**. The external view determines what subset of geometry from your model will be analyzed.

For example, if you created a solid model and extracted a mid-surface, you would assign the mid-surface to the external view. To create an external view, select External View from the **Tools** menu in the workbench. The dialog shown at left will appear. Select the geometry from the model and press **OK**.

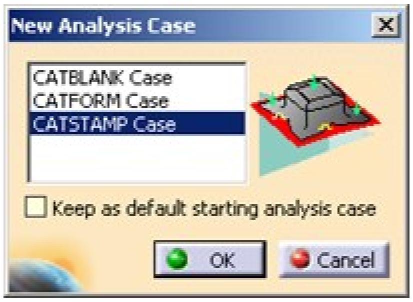
Chapter 3: Setting Up the Analysis

In this section:

- Starting the Analysis
- Defining the Material
- Defining the Mesh Parameters
- Curved Binder
- Setting the Material Thickness
- Tipping and Symmetry
- Forming Process
- Friction and Blankholder Force
- Pressure Pads
- Draw Beads
- Target Strain
- Clamps
- User Defined Restraint
- Enforced displacement
- Solving

Starting the Analysis

When you have completed the steps outlined in Chapter 2 and you are ready to start your analysis, select **FTI Forming Analysis** from the **Analysis and Simulation** section of the CATIA **Start** menu.



A dialog will appear prompting the user to select an analysis case. Available options may vary according to the license held by the user. Selecting the highest level option (CATSTAMP) will give you the most functionality and complexity whereas the lower level options (CATBLANK) will provide a simpler interface requiring fewer inputs but with less flexibility and power.

Note: It is recommended that the **Keep as default** starting analysis case option is not selected. When this option is selected the initial part mesh settings are typically not suitable for forming analysis and will need to be modified before the part mesh is generated.

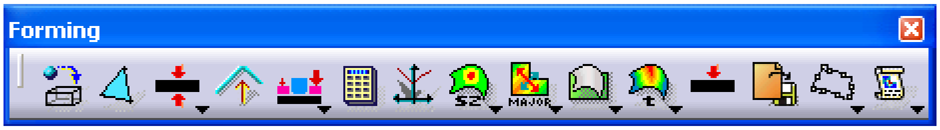
After selecting an analysis case, a new Analysis workbench will be created and a new category called **Finite Element Model.1** will be added to the tree (see below). Under this branch will be entries for the mesh, inputs and the **CATSTAMP (CATFORM or CATBLANK) Forming Solution Set** where the result plots will be stored.

Note: If part of the analysis tree is missing, it is likely that you are experiencing licensing problems. You may be missing the license, using the wrong license or all of the licenses (on a network license) may currently be checked out.

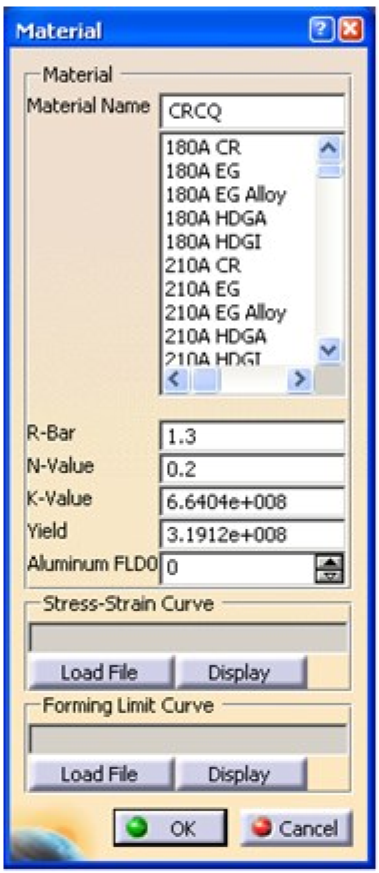
Defining the Material

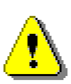
Because the standard CATIA material library does not support the material properties required to run a forming analysis, the material must be re-defined in the analysis workbench. This also enables us to model different materials in different areas of the part to represent a tailor welded blank.

To define a material, select the **Material Selection** icon from the **Forming** toolbar (see below).



Select a material from the list. Check the values below to ensure that they are correct. Sometimes, these properties will vary from supplier to supplier.



 **Important!** If you are using any material other than steel (ex. aluminum), make sure you enter in an FLD0 value near the bottom of the dialog. This value will be calculated from the other material properties if it is set to zero, however the calculation is only valid for ferrous metals (steel, iron, etc). The FLD0 value determines the point at which the material will fail so it is critical to obtaining useful analysis results.

At the bottom of the dialog, you can optionally load a stress strain curve for the material. This will typically improve results for materials with unique stress/strain relationships (ex. dual phase steel, etc.).

If the part will be made from a tailor welded blank, assign one material for each section of the part.

The units for the stress-strain curve are psi for stress and true (decimal) for strain. The stress-strain curve files are text files (.txt) with the extensions renamed to .ssc. The format of the files is two columns, with strain in the first and stress in the second. The values are separated by a space.

The units for the forming limit curve are engineering %. The forming limit curve files are text files (.txt) with the extensions renamed to .flc. The format of the files is two columns with minor strain in the first and major strain in the second. The values should be separated by one or more spaces.

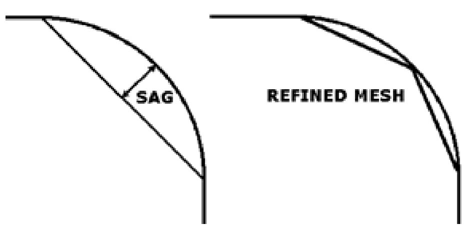
Defining the Mesh Parameters

In this manual we will cover mesh generation using CATIA's **Octree Triangle Mesher** tools. If you have the license for the **Advanced Meshing Tools** workbench, you can also use that to generate your mesh however the Octree Triangle Mesher is simpler to use and usually yields more than adequate results.

When an analysis is created, the mesh part will automatically be assigned to the part defined using the External View command (see Section 2.8). You can change the mesh size settings by double clicking on the green triangle on the part or by editing the definition of the **Part Mesh** entry in the tree (located under **Nodes and Elements**).

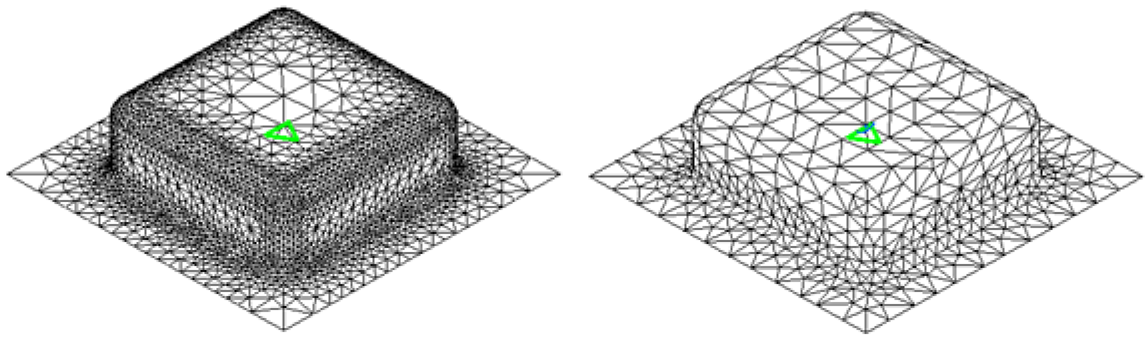
The dialog will appear with default values in the three text boxes.

- The **Size** value determines the maximum size of mesh element that will be created. This value should typically lie in the range of 10 to 50mm.
- The **Absolute Sag** value determines the allowable deviation from the true geometry. In areas where the geometry is curved, the elements (which are all flat) will not deviate from the true curve more than this value (see below). If the deviation is higher, the mesh will be automatically broken down (refined) into more, smaller elements during the meshing process. This value should typically be set between 0.1 to 0.5mm.
- **Proportional Sag** is the ratio between the local absolute sag and the local mesh edge length. The default value is 0.2



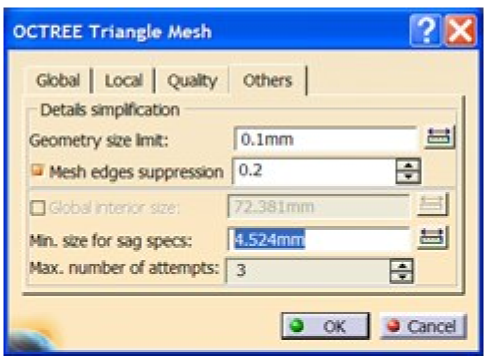
NOTE: The most constraining of the Absolute Sag and Proportional Sag values will be used during mesh generation.

A good mesh will typically have at least three elements on the critical radii (below left). Fewer elements (below right) will not simulate the material flow as accurately. If this is the case, try using a smaller sag value to create more elements. Notice that large elements are just as effective at modeling flat areas as smaller elements.



GOOD MESH POOR MESH

If you have licensing for the advanced meshing tools, you can reduce the minimum element size by changing the minimum size for sag specs on the last tab (Others) in the meshing dialog (shown below).



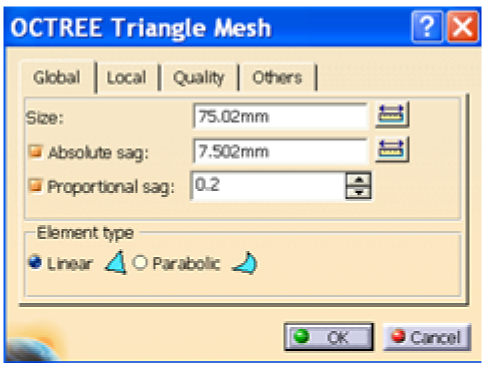
NOTE: Using a smaller size or sag value will create more mesh elements in your model. Increasing the number of mesh elements will increase the time required to solve the part. Care should be taken to ensure that an excessive number of mesh elements is not used.

Curved Binder

By default, the FTICATIA analysis is performed with a default flat binder. In many cases, the actual forming starts with a curved binder that creates a curved blank. The curved blank usually results in more material being available for the forming operation. This extra material has a significant effect on the results depending on the forming conditions that were applied.

To run a curved binder analysis you must have a surface representing the binder geometry. The binder shape must be developable and must be large enough to contain the developed blank shape for the model. A developable binder is defined as one that has been bent into a simple curve in such a way that if it were flattened, there would be little or no residual strain in the blank. A separate mesh must be created for the curved binder geometry.

To create a mesh for the curved binder select the Octree Triangular Mesher icon from the forming toolbar and select the curved binder geometry. When meshing this binder, the mesh does not have to relate to the mesh size of the part.



Important!: Curved Binder function is not available with CATFORM

Setting the Material Thickness

The next step in the analysis procedure is to set the material thickness. Because the model being analyzed is a surface model with effectively zero thickness, the thickness must be set prior to analysis.

When you start an analysis, a default thickness values and direction is automatically assigned to the model. This thickness value can be seen as a number highlighted on top of the geometry.

To change the material thickness value, double click on the number or edit the definition of **2D Property.1** in the tree under **Properties.1**.

The thickness dialog will appear. Change the thickness value to the correct thickness.

Note: to ensure that the thickness value is being used by the analysis corresponds to the thickness value in the model; it is a good idea to use a formula to link the thickness value with the part geometry.

To change thickness direction, double click on the arrow sign or edit the definition of Thickness Direction in the tree under **Restraint Set**.

If you are conducting an analysis on a part made with a **tailor welded blank**, select one material type for each segment of the part using the **Material Selection** icon (even if two segments are the same material, there must be two definitions). The first material and the default thickness value will automatically be assigned to the whole model. To assign additional materials and thicknesses to different regions on the part, press the **Material Thickness** icon (see below) from the **Forming toolbar**.

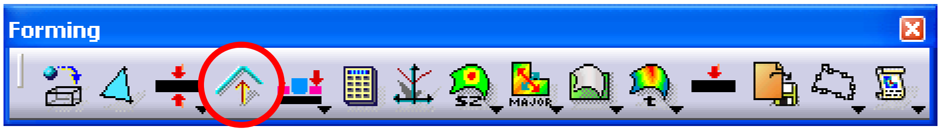
Select the area of the part to assign the second material and thickness to and enter the thickness. Repeat this procedure until a thickness has been assigned to all regions of the model.

NOTE: You can only assign the second thickness when you have a CATFORM or CATSTAMP license

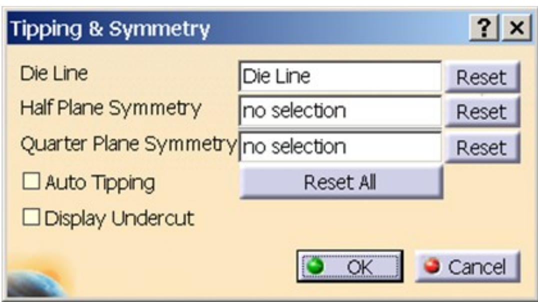
Tipping and Symmetry

Most CATIA models are constructed in the assembly position. This is usually not the position the part will be formed in. For this reason, we must define the press direction prior to running the analysis.

To set the press direction (or "tip" the part), select the **Tipping & Symmetry** icon (see below) from the **Forming** toolbar.



If you do not know the forming direction and you want the software to automatically determine an approximate orientation, check the **Auto Tipping** check box and close the dialog.



Alternatively, you can create a line or a planar surface in your part model which can be used to set the press direction. Click on the line or the planar surface and close the dialog.

Symmetry

Sometimes it is desirable to model only half of a part to speed up model creation and solution time. FTICATIA has the ability to model only half of a symmetrical part by applying a symmetry condition that will model the part as if the other half was also present.

To apply symmetry, create a plane in the part model that represents the plane of symmetry. Then, in the tipping and symmetry dialog, select the plane. After defining the symmetry plane you must also define the tip direction using either a line or planar surface. You can also apply a second symmetry plane to model only a quarter of a part.

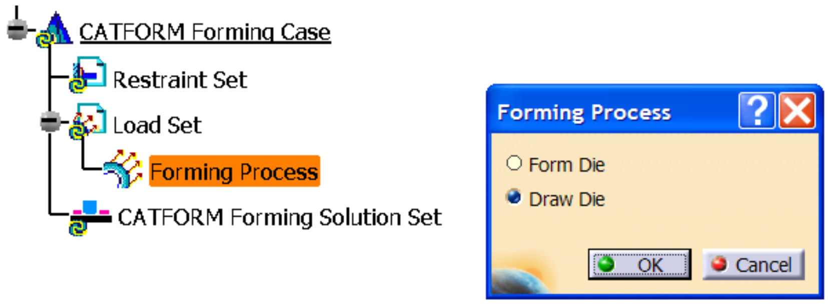
Undercut

To display the elements which are undercut, select the Display Undercut button. The model will need to be meshed prior to display of the undercut elements. If there are undercut areas in the model, use the tipping functions to try to eliminate the undercut areas as much as possible.

Forming Process

This option allows you to input the type of process that you are using to manufacture the part. This will influence the way the part deforms during forming and can affect the blank shape. You can choose between a Draw Die and a Form Die.

The forming process is set to Draw Die by default. If you want to change between the Draw Die and the Form Die process, double click on the **Forming Process** branch in the Specification Tree to open the dialog is shown below:

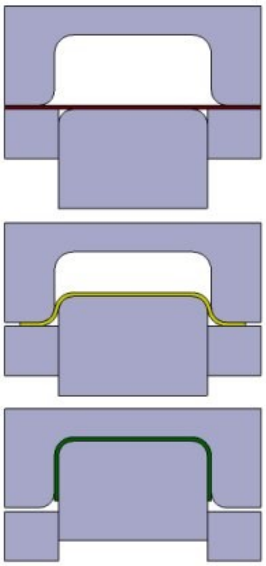


Draw Die

If you are forming a part using a draw die (illustrated below left) you should use the Draw Die forming process.

As material passes over the radius on the die, additional strain is added to the part due to the friction between the part and the die and the bending and unbending process. These additional strains are not present when a part is manufactured using a form die.

The friction and bending effects are automatically considered by FormingSuite when the Draw Die setting is turned on. The Draw Die process formulation determines the areas of curvature on the part and adds the required forces and strains to any material that would flow over them during forming.

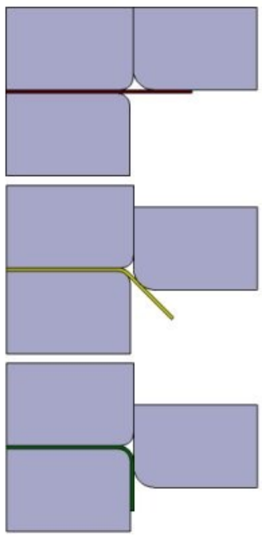


Draw Die

Form Die

If you are forming a part using a Form Die (illustrated below right) you should use the Form Die forming process. This process is typically used in progressive dies and simple flanging operations.

In this type of operation, the blank material is not pulled through a radius; it is simply bent around it. As a result, there are no additional strains produced by friction.



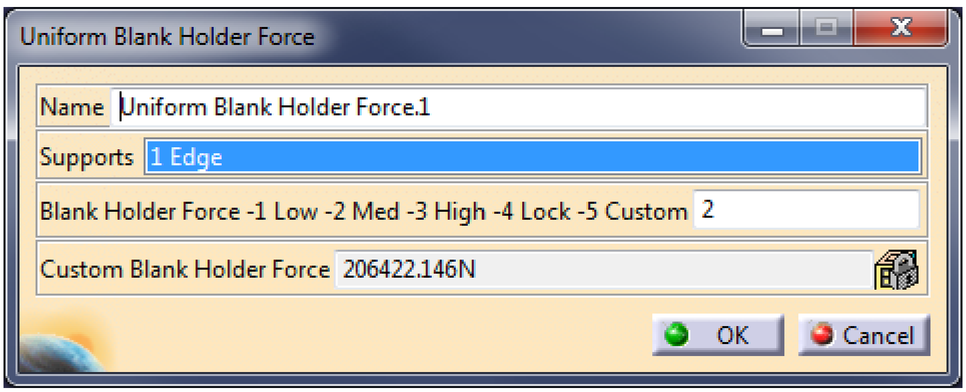
Form Die

Friction and Blankholder Force

CATFORM and CATSTAMP have the ability to simulate the effects of a uniformly distributed blankholder force. It does this by applying a uniform edge tension to the boundary of the part during simulation. This will lead to additional stretch in the model and will help reduce wrinkling.

Additionally, you can modify the friction coefficient used during the analysis to simulate special lubricants or coatings.

To apply a blankholder force to your part, select the **Friction and Blank Holder** icon (see below) from the **Process Condition** toolbar.



To apply a blankholder force, specify a value for the force in the **BHF** text box. 1 is used to simulate a low holding force, 2 is medium, 3 is high and 4 will lock the edges of the part so it is formed as pure stretch. The actual forces that correspond with these settings vary based on the material properties. You can also set the blankholder force to a specific value by putting a 5 in the **BHF** text box and entering the force in the **User Defined Blank Holder Force** text box.

You can apply local blankholder forces using the **Variable Blank Holder** icon in the **Process Condition** toolbar.

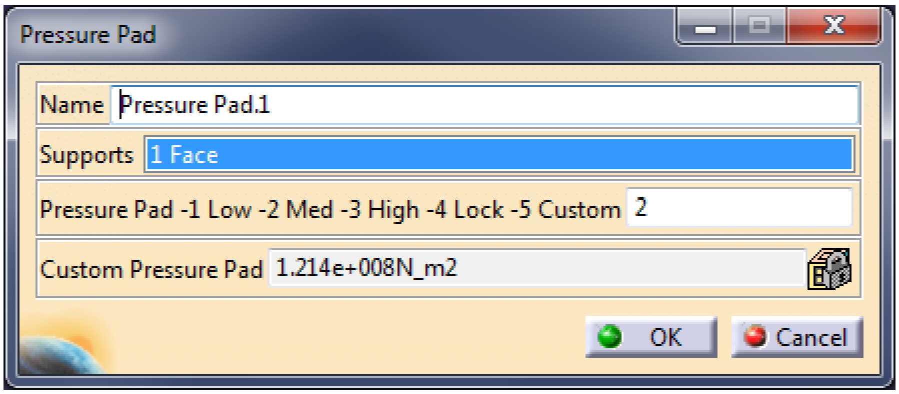
Select lines/curves on the edge(s) of the part to apply the blankholder to and specify the force. You can add multiple unique blankholder forces in different areas.

NOTE: Blankholder Force functions are not available with CATBLANK

Pressure Pads

CATFORM and CATSTAMP can also simulate the effects of one or more pressure pads that may be applied to the part during forming. It does this by restraining the movement of the mesh elements in the area under the pad.

To apply a pressure pad to your part, select the **Pressure Pad** icon (see below) from the **Process Condition** toolbar.



To apply a pressure pad, select the face(s) to constrain from the part model. The entire surface of the selected face(s) will be constrained by the simulated pad. If you want to apply the pad to only part of the face, split the face using the CATIA geometry tools.

After selecting the faces, specify a value for the pad pressure in the **Pad Pressure** text box. Note that this is the pressure the pad is exerting on the part and not the cylinder pressure.

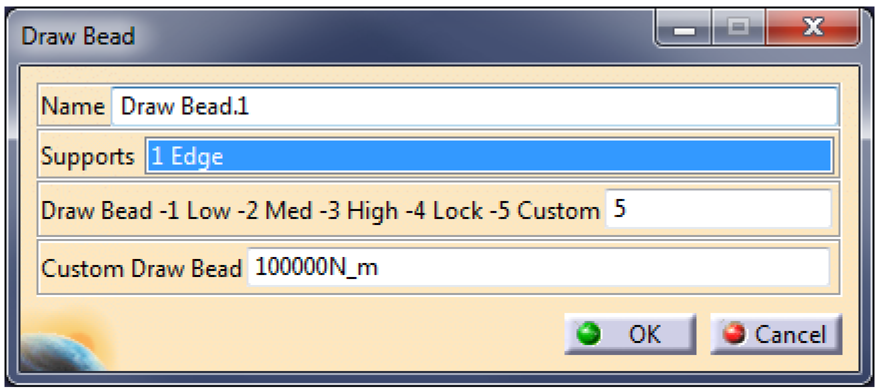
Tip: It is generally best to run the first analysis on a part with no pressure pads or blankholder force since adding these forces increases the stretch. If a part splits with no forces, it will almost certainly split if they are added.

NOTE: Pressure Pads function is not available with CATBLANK.

Draw Beads

This forming condition can be used to simulate the control of material flow into the die cavity using a draw bead or a lock bead.

To apply a draw bead to your part, select the **Draw Bead** icon (see below) from the **Process Condition** toolbar.



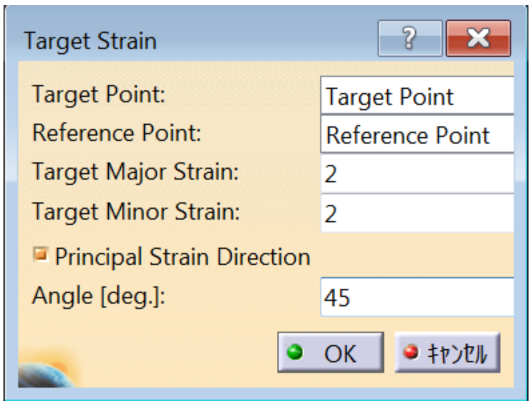
To apply a draw bead, select the geometry representing the location (Supports) and specify a value for the Draw Bead Force. Geometry from a sketch cannot be selected. Geometry on a sketch must first be projected on to the part before it can be used to define the location of a draw bead. All other splines, polylines, lines and arcs can be used.

NOTE: Draw Bead function is only available for CATSTAMP.

Target Strain

Target Strain is used when analyzing class ‘A’ product geometry (without flanges/hems or addendum) to simulate an ideal dieface design that would deliver a defined strain pattern at a specified location. It may not be possible to design physical tooling that will yield the same results, therefore Target Strain results should be reviewed with this in mind.

To apply Target Strain to your part, select the **Target Strain** icon (see below) from the **Process Condition** toolbar.



To apply Target Strain to a part, select the point on the geometry representing the desired target strain location (Target Point).

Select the point of first Contact where the punch first touches the blank (Reference Point). This step is optional; it is only necessary if the Point of First Contact is different than the Target Point.

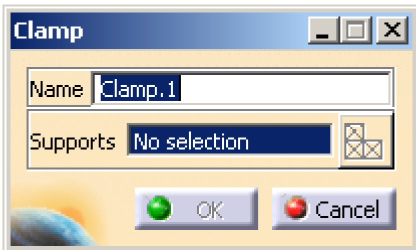
Input both Major and Minor strains (Target Major Strain & Target Minor Strain) in percent engineering strain.

Check the Principal Strain Direction checkbox to specify the major strain direction as an angle (in degrees) measured from the x-axis of the axis system used for tipping. This step is optional. It is helpful to define the tip direction using a coordinate system rather than using either auto-tipping or a single axis die line since a tip direction defined by a coordinate system defines a clear x-axis for the AutoStrain calculations. Using auto-tipping or a single axis die line does not explicitly define x-axis; this may produce unexpected results.

Clamps

All FTICATIA analysis products allow the user to fix the position of a face, edge or point on the X-Y plane between the blank and part by defining a clamp.

To apply a clamp to your part, select the **Clamp** icon (see below) from the **Process Condition** toolbar.

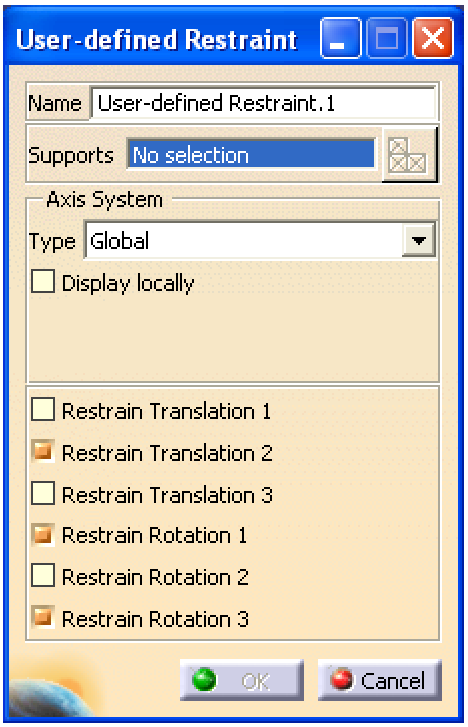


To apply a clamp, select a face, edge and/or vertex of the part geometry representing the location (Supports) for the clamp.

User Defined Restraint

All FTICATIA analysis products allow the user to fix any combination of available nodal degree of freedom on a face, edge or point by defining a **User Defined Restraint**. There are three translation degrees of freedom and three rotation degrees of freedom that could be restrained

To apply a **User Defined Restraint** to your part, select the **User Defined Restraint** icon (see below) from the **Process Condition** toolbar.



To apply a user defined restraint, select a face, edge and/or vertex of the part geometry representing the location (Supports) for the user defined restraint.

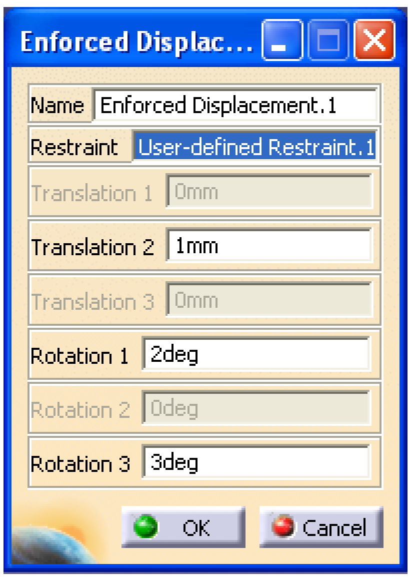
Enforced Displacement

Enforced Displacements are loads applied to support geometries, resulting for the subsequent analysis in assigning non-zero values to displacements in previously restrained directions.

Enforced Displacement objects belong to Loads objects sets. An Enforced Displacement object is by definition associated to a Restraint object.

Make sure you entered non-zero values only for those degrees of freedom which have been fixed by the associated Restraint object. Non-zero values for any other degree of freedom will be ignored by the program

To apply an Enforced Displacements to your part, select the **Enforced Displacement** icon (see below) from the **Process Condition** toolbar.



To apply an enforced displacement, select the restraint previously defined by User defined restraint, and then apply a non-zero number to the directions defined by User Defined Restraint.

Friction

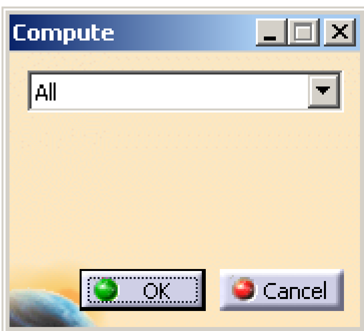
The friction coefficient used in the analysis is set in the Friction object in the Restraint Set in the feature tree. This object is automatically created when you create an analysis. The default value of 0.15 represents a typical, conservative value for mill oil on steel. You can change this value to a lower value when a lubricant is being used or when tooling has special, anti-friction coatings. Conversely, it can be helpful to raise the value slightly when forming aluminum or increasing it substantially (up to 0.3) for simulating hot forming.

Solving

When the model set up is complete, the final step before viewing the results is meshing and solving. In CATIAV5, this is combined into one step.

Select the **Mesh & Compute** icon (see below) from the **Forming** toolbar.

To start the analysis, set the drop-down box to **All** and press **OK**. This will first mesh the part and then run the solver. To generate the mesh without solving set the option to **Mesh Only** and press **OK**.



Meshing will typically take a few seconds to a couple of minutes. The solving process could take just a few seconds to several minutes for a complex part.

On occasion, the mesh will fail. This may be caused by problems with the geometry. The problem can often be overcome by simply changing to a slightly different mesh size and re-running the solution.

Once you have generated results, any changes to the input geometry or analysis setup will cause the solver and the mesher (if required) to re-run when you try to re-activate a result plot. This update mechanism ensures that the analysis results are always consistent with the current model.

Chapter 4: Viewing Results

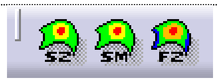
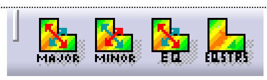
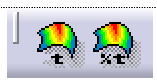
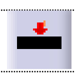
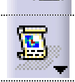
In this section:

- Generating a Result
- Results
- Forming Limit Diagram
- Create Blank Face Feature
- Point Mapping
- Part Weight Calculator
- Re-Running the Analysis After Changing the Inputs
- Multiple Forming Cases
- Typical Analysis Methodology

Generating a Result

FTICATIA provides a comprehensive set of graphical results plots along with a punch force calculation and several blank shape generation tools. These results can be used to quickly find hot spots, create cost estimates and create an in depth formability analysis of the part.

Select the Forming Solution Set from the feature tree to specify the analysis for which you want to generate a result plot. Then, select the desired result plot icon from the forming toolbar (see below).

	▪ Part and Blank ▪ 3D and 2D Displacement ▪ X, Y and Z Displacement
	▪ Create Blank Face ▪ Point Mapping ▪ Part Weight
	▪ Safety Zones ▪ Safety Margin ▪ Forming Zones
	▪ Major Strain ▪ Minor Strain ▪ Equivalent Strain ▪ Equivalent Stress
	▪ Thickness ▪ Thickness Strain
	▪ Punch Force
	▪ Export Results
	▪ Basis Analysis Report

NOTE: When viewing multiple result plots, make sure you deactivate (not hide) all the result plots other than the current one. Otherwise, the results will overlap on the screen. You may also have to hide the mesh (Nodes and Elements) and the original geometry (Links Manager) in order to view the result plots clearly.

Results

Blank Shape

The blank shape plot displays the blank (deformed mesh) on the screen underneath (or over) the part mesh. Since CATSTAMP simulates the forming in reverse (i.e. from the part to the blank) the deformed mesh will be the flat pattern for the part.

Forming Displacement

Displays the various displacement components as calculated between the part and blank.

Direction	Description
3D	The total node displacement between the part and blank. The square root of the X displacement squared plus the Y displacement squared plus the Z displacement squared.
2D	Nodal displacement between the part and the blank in the X-Y plane.
X	Node displacement in the X direction between the part and blank.
Y	Node displacement in the Y direction between the part and blank.
Z	Node displacement in the Z direction between the part and blank.

NOTE: Forming Displacement plots are only available for CATSTAMP

Create Blank Face

To convert the blank to an associative surface select the **Create Blank Face** icon from the **Forming** toolbar.

Point Mapping

To map a point from the part to the blank shape, select the **Point Mapping** icon on the **Forming** toolbar.

Part Weight

To calculate the part weight, accounting for the change in material thickness during the forming operation, select **Part Weight**. You will be prompted to select the geometry to calculate the part weight for. The geometry selected does not have to be the geometry defined as the external view for the analysis.

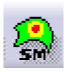
Safety Zone

The safety zone plot shows whether or not a part will be formable. It is a qualitative plot (shows trends, not numbers). Each color represents a different zone. These zones are described in the table below:

Value	Zone	Description
0-1	Strong Wrinkling Tendency	Material is gathering and thickening. Stresses are in compression in all in-plane directions. If the material is unsupported, it will likely wrinkle in this area.
1-2	Mild Wrinkling Tendency	Material is gathering and thickening somewhat. Stresses are in compression in the minor strain direction. Wrinkling may occur in this area.
2-3	Safe	Material will not split or wrinkle but thinning could be higher than desirable.
3-4	Marginal	The part is very close to splitting in this area. Any problems with the process will create a split. If all things are perfect, the part may form.
4-5	Failure	The part is being stretched too much and will split in this area unless special considerations are made during tooling design.

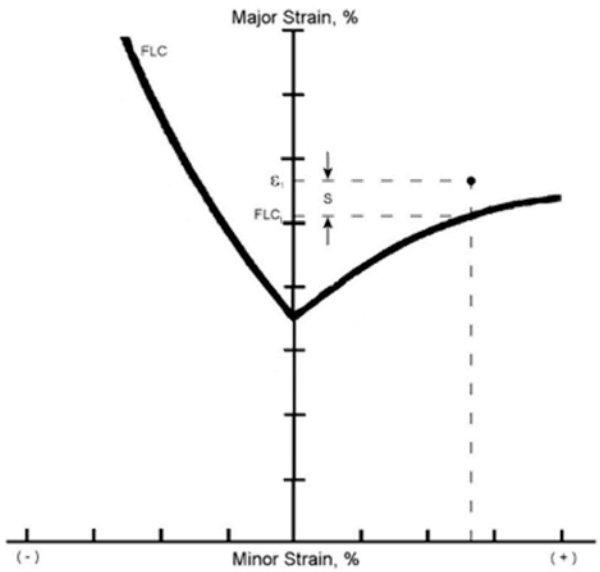
NOTE: Safety Zone function is not available for CATBLANK

Safety Margin



Displays the relative strain between the FLC and major strain value at each node.

- S = 1, the major strain of the nodal point is 0
- S > 0, the nodal point is below the FLC and may be safe
- S = 0, the nodal point lies on the FLC



$$SafetyMargin(S) = \frac{(FLC_1 - \varepsilon_1)}{FLC_1}$$

NOTE: Safety Margin function is only available for CATSTAMP

Forming Zone

The Forming Zone plot describes the state of stretch in the part. It is a qualitative plot (shows trends, not numbers). Each color represents a different zone. These zones are described in the table below:

Value	Zone	Description
0-1	Strong Wrinkling Tendency	Material is gathering and thickening. Stresses are in compression in all in-plane directions. If the material is unsupported, it will likely wrinkle in this area.
1-2	Mild Wrinkling Tendency	Material is gathering and thickening somewhat. Stresses are in compression in the minor strain direction. Wrinkling may occur in this area.
2-3	Loose Material	Material is unsupported and could lead to poor surface effects such as oil canning.
3-4	Semi-Tight Panel	Material is being stretched but minor strain is negative.
4-5	Plane Strain	Material is experiencing stretch in only major strain direction.
5-6	Tight Panel	Material is being stretched in both major strain and minor strain directions. This leads to good stiffness and dent resistance but may also cause high thinning.

NOTE: Forming Zone function is not available for CATBLANK

Major Strain

The major strain is measured at each node on the mesh (each corner of each element). It is defined as the strain (% stretch) in whatever direction has the greatest in-plane stretch.

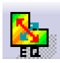
NOTE: Major Strain function is not available for CATBLANK

Minor Strain

The minor strain is the strain in the in-plane direction perpendicular to the direction of the major strain.

NOTE: Minor Strain function is not available for CATBLANK.

Equivalent Strain



The hypothetical strain representing the sum of the Thickness, Major and Minor strains values in a uniaxial tension test.

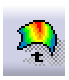
NOTE: Equivalent Strain function is only available for CATSTAMP

Equivalent Stress

The hypothetical stress required to deform the material by the equivalent strain amount in a uniaxial tension test.

NOTE: Equivalent Stress function is only available for CATSTAMP

Thickness



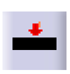
This plot shows the actual material thickness after the forming operation.

Thickness Strain

This plot shows the percent change in material thickness resulting from the forming process. It is often easier to interpret than the Thickness plot since determining the change does not depend on the original thickness of the blanks.

NOTE: Thickness Strain function is not available for CATBLANK.

Punch Force



Displays the force required to deform the material.

NOTE: Punch Force function is only available for CATSTAMP.

Export Results

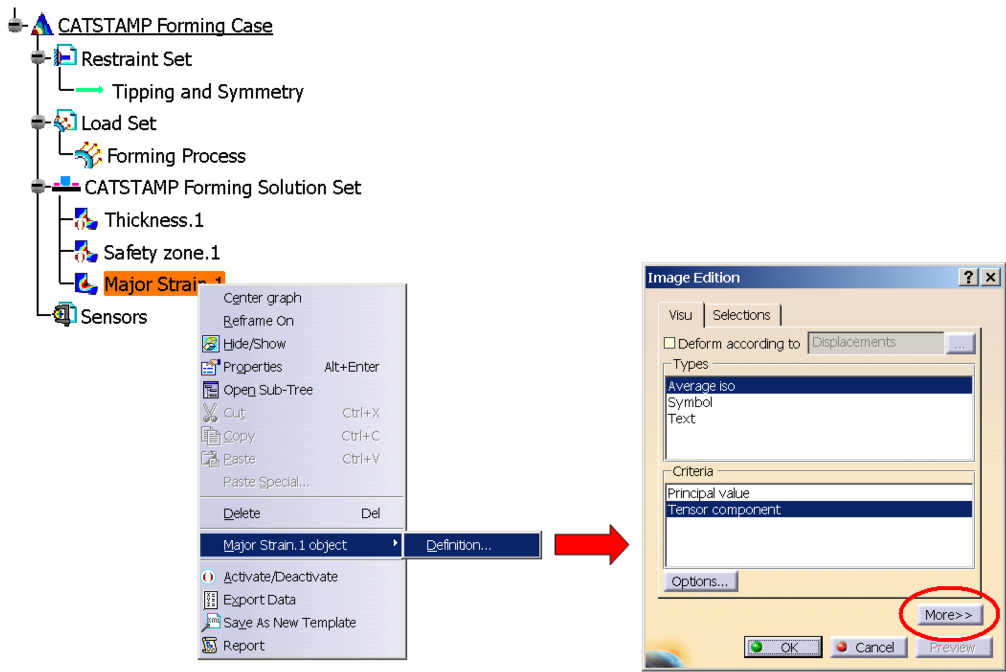
Export thickness and strain results for downstream CAE applications as a Nastran or LS-DYNA file.

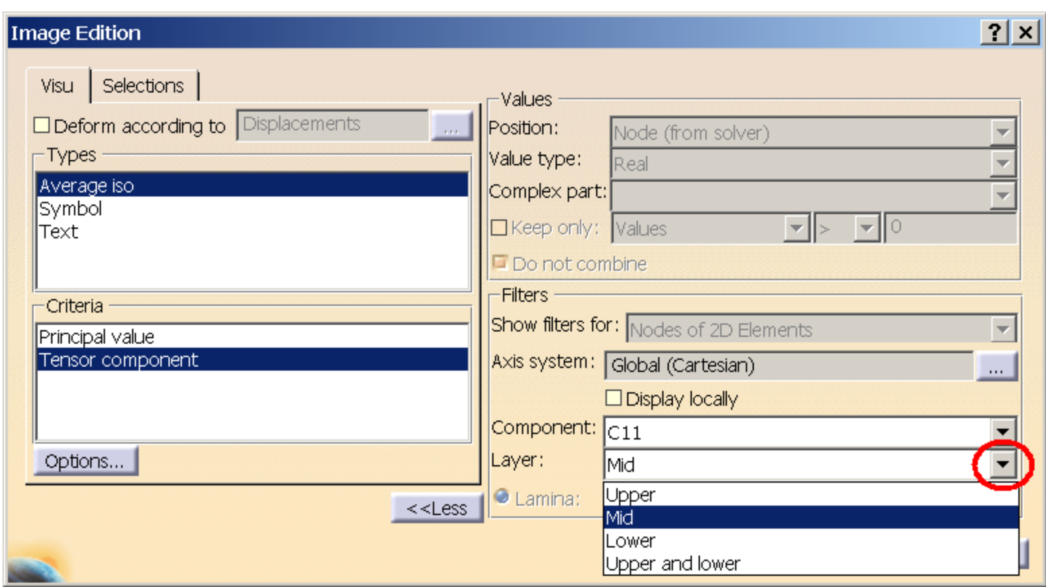
NOTE: Exporting Results function is not available for CATBLANK.

Displaying Results for Top, Mid, or Bottom Surfaces

By default, all results are shown for the mid-plane of the model. Certain result plots, namely; Safety Margin, Major Strain, Minor Strain, Equivalent Strain, and Equivalent Stress can be configured to display top, middle, or bottom results. The remaining result plots will only display the mid-plane results.

To change whether you are displaying top, middle, or bottom results, you must right-click on the result plot from the specification tree and select definition from the menu to open the Image Edition dialog (shown below). Once the dialog is open expand it by selecting the More button, then select the drop down menu for Layer and choose either upper, mid, or lower.



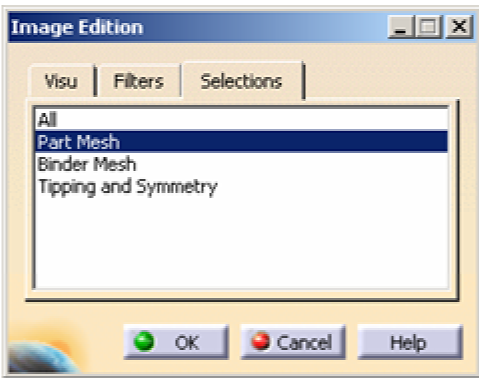


Basic Analysis Report

Create an HTML based report summarizing the formability analysis.

Important Note: When viewing results, always make sure you are using the Custom view mode with the Material option turned on. Otherwise, most results will not appear correctly. You can modify the render style by selecting Render Style from the View menu and selecting Customize View from the sub-menu.

TIP: To hide the curved binder geometry in a plot, edit the definition of the plot and turn on the part geometry filter. To edit the definition of the plot, right click on the result plot in the Specifications tree and select **<result plot name> object -> Definition**.



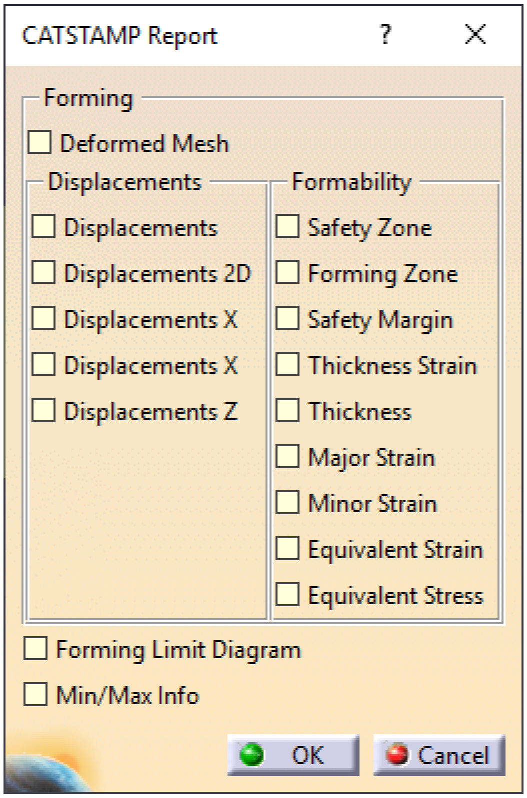
Then on the **Selections** tab click on **Part Mesh** and select **OK**.

FTI Forming Reports

Once you have completed your formability analysis, you can export a report.

To create a CATSTAMP report

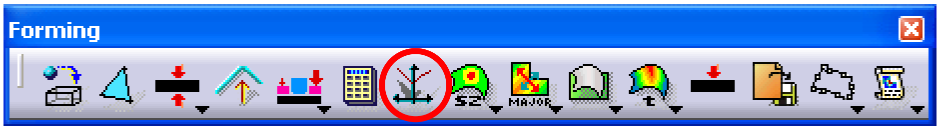
1. On the feature tree, right-click on **FTICATSTAMP** and then click **FTICATSTAMP Object** > **Report**.



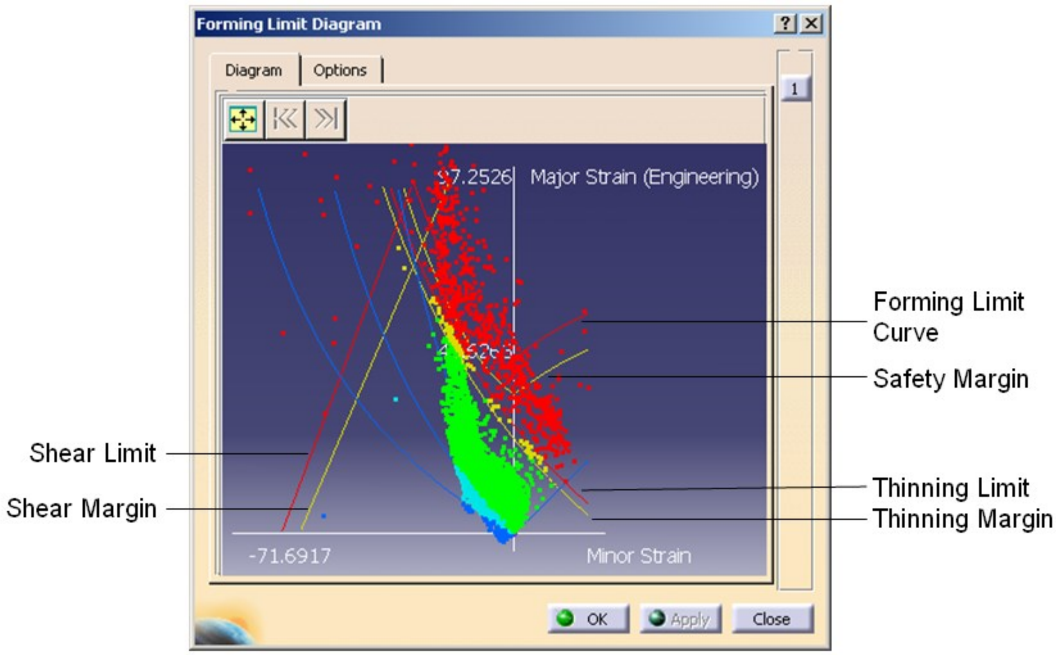
2. Click to select any Formability plots that you want to include in the report.
3. Click **Forming Limit Diagram** to include an FLD plot.
4. Click **Min/Max Info** to include custom min and max points in the report.
5. Click **OK** to generate the report.

Forming Limit Diagram

The strain distribution for all the nodes on the part can be displayed on a Forming Limit Diagram (FLD). The FLD can be used to determine the forming mode and the formability of the part in a local area.



To display the FLD result, select the Forming Limit Diagram icon on the Forming toolbar.



TIP: If additional material types or thickness have been defined for a Tailor Welded Blank then separate FLD's will be created for each material. Click the numbered tabs on the right-hand side of the Forming Limit Diagram window to display the selected FLD and edit its options.

The diagram and safety limits for the FLD can be changed from the Options tab. Change any field then return to the Diagram tab, the settings will be applied and the FLD updated automatically.

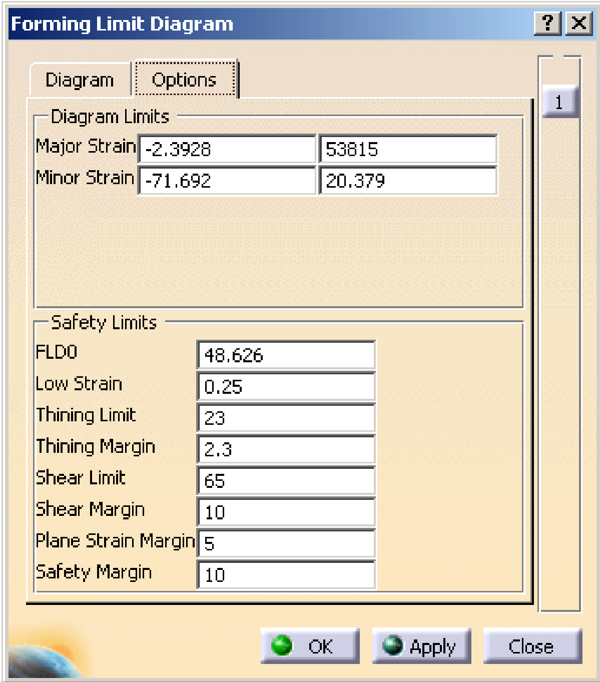

Forming Limit Diagram
Diagram
Options
Diagram Limits
Major Strain -2.3928 53815
Minor Strain -71.692 20.379
Safety Limits
FLD0 48.626
Low Strain 0.25
Thining Limit 23
Thining Margin 2.3
Shear Limit 65
Shear Margin 10
Plane Strain Margin 5
Safety Margin 10
OK
Apply
Close

Diagram Limits

The default Major and Minor Strain axis limits are determined from the maximum and minimum major and minor strains. To change the axis limits enter a new value in the desired field.

Note: Closing and reopening the FLD will restore the default axis limits.

FLD0

FLD0 is the major strain value where the forming limit curve crosses the major strain axis (minor strain equals zero). There are two possible default settings for FLD0, if FLD0 is defined as a material property (non-zero value) then this value is used. Otherwise, if it is set to zero as a material property the FLD0 will be calculated from the n-value and material thickness. To change the default FLD0 enter a new value into the FLD0 field.

Low Strain Zone

A Low Strain zone can be defined for the Safety Zone plot. The Low Strain zone is a circular area centered on the origin of the Forming Limit Diagram (0% major and minor strain). The default value is 0.25.

Thinning and Shear Limits

Thinning and Shear Limits can be defined for the Safety Zone plot. The red line on the right-hand side of the plot is the Thinning Limit Line. The red line on the left-hand side represents the Shear Limit Line. The default values are 23 and 65 for Thinning and Shear respectively.

Thinning and Shear Margins

Thinning and Shear Margins can be defined for the Safety Zone plot. The yellow line on the right-hand side of the plot is the Thinning Margin Line. The yellow line on the left-hand side represents the Shear Margin Line. The default values are 2.3 and 6.5 for Thinning and Shear respectively, 10% of the limit values.

Plane Strain Margin

The default Plane Strain Margin defining the plane strain zone is set to the area on the FLD where the minor strain is 5% of the major strain. To change this margin, change the setting in the Plane Strain Margin field.

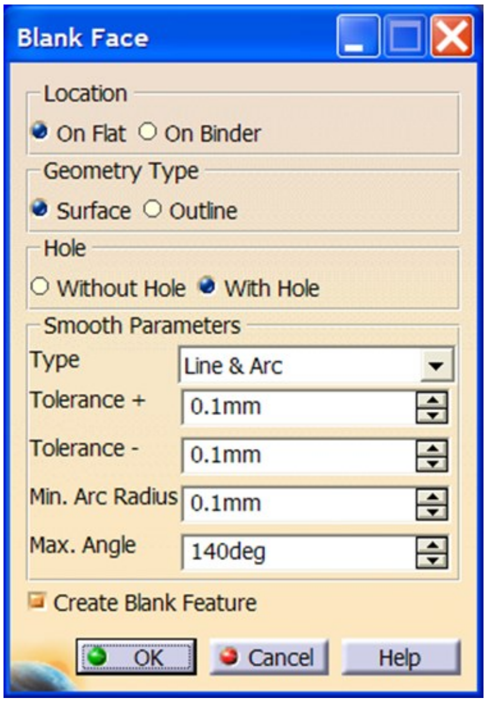
Safety Margin

The default Safety Margin defining the marginal splitting zone is set to the area between the Forming Limit Curve and the safety curve, which is offset below by 10%. To change this offset, change the setting in the Safety Margin field.

NOTE: A safety margin of 10% is typical for steels. For Aluminum a typical safety margin is 6 to 7% The Forming Limit Diagram is not available for CATBLANK

Create Blank Face Feature

To create the blank face geometry, select the Create Blank Face Feature icon from the blank face toolbar. This will open the dialog, which will contain various options depending on the type of analysis run.



If you ran a curved binder analysis, you will have the option to create the blank on the binder surface as well. If you select the location on binder, you will only be able to select the outline option for geometry type.

If the part geometry in the analysis contained any holes, you will have the option to create the blank with or without these holes included.

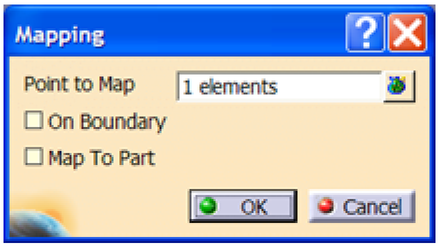
The smoothing parameters allow you to choose how you would like to smooth the blank boundary. You choose from Line & Arc, B-Spline, Line, and None. If you choose Line and Arc, you will need to specify the tolerance, min. arc radius, and the max. angle. The smoothing type none, refers to leaving the boundary of the blank to be made up of the boundary mesh element edges.

The option to Create Blank Feature allows for associatively between the blank face feature and the analysis. If this option is un-checked, then any changes to the analysis after the blank is created will not be reflected in the blank geometry.

Point Mapping

The purpose of this function is to provide a tool that allows you to map points, lines, and edges from the part model to the blank and vice versa.

To map entities from the part to the blank open the point mapping function after solving, select the entities that you would like to map, and click OK. To map entities from the blank to the part, you will need to check the Map To Part option.



If you performed an analysis using a curved binder, you will also have the option to **Map To Binder**.

Part Weight Calculator

The purpose of this function is to provide a very accurate part weight that considers the following:

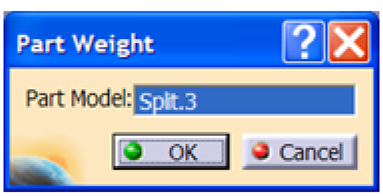
- Thinning and thickening that occurs during the forming process, &
- Holes that are removed after forming the part

This function is meant to replace the current practice of basing the weight on the volume of a uniform thickness solid model, which typically leads to over-estimates of part weight.

To display the value in the specification tree, you must choose to display the value under Options > General > Parameters and Measure as well as Options > Analysis & Simulation > General.

Procedure:

1. A forming analysis is performed on a version of the part geometry with holes filled and possibly addendum added. This is generally the draw geometry.
2. The blank shape is calculated from the above forming analysis.
3. You then select the model of the part to use for mapping/trimming and hole removal. You will first need to show the final part geometry in the GSD workbench and hide the draw geometry.
4. Open the Part Weight Calculator and select the final part geometry.



5. Thickness and mapping information is taken from the current solution set
6. A new blank is created in the feature tree – it is made up of an outline (created by mapping the part boundary to the analysis blank), a filled surface (to create the new blank face), another mapping and a join (to define the holes) and a split to remove the holes from the blank face
7. The weight will be calculated from the surface area of the split and is displayed as a parameter on the Specifications Tree in the Analysis Workbench.

Re-Running the Analysis after Changing the Inputs

If you make changes to any of the analysis inputs that will affect the results (ex. geometry, mesh size, material, etc.) all result plots will be deactivated. When you reactivate a result plot, you will be warned that the analysis results are out of date and you will be asked if you want to proceed. Clicking yes will re-mesh the model if necessary and re-run the solution with the new input values. When the solution is complete, the result plots can then be viewed again.

This parametric link between the solution and all the inputs will prevent any user from viewing data that is no longer valid due to a change implemented between the first time the analysis was run and the point when the user reopens the result for viewing.

Multiple Forming Cases

Multiple Forming Cases can be analyzed at once to compare the effects of different settings on the analysis results. Unique tipping positions and forming constraints can be applied to each forming case.

Select **Forming Case** from the **Insert** menu to add a second forming case.

Each forming case will be listed as a unique branch in the Specifications tree. The active forming case will be underlined. To view the results from the non-active forming case, right click on the forming case and select Set As Current Case.

Typical Analysis Methodology

When analyzing product geometry, the main concern is identifying potential formability issues. This usually involves a qualitative approach which means that the numerical results are of less importance than the trends. There are a number of variables that can be changed in the tool to influence the strain values but geometrical problems such as radii and wrinkling are more difficult to remedy. To identify these problems, start by analyzing the part with no blankholder force.

If the part splits inside the part, you will need to change the geometry or material. A split on the part edge may be remedied by adding extra material during forming and trimming later so this is not as significant. Increasing the blankholder force at this point would simply exaggerate the splitting problem.

If the part does not exhibit any splitting or excess thinning but does show some wrinkling, increase the blankholder force (one step at a time) and re-run the analysis. If the new results show splitting and wrinkling is still a problem, some geometry changes may be required. If there are still no splits and the wrinkling problem has been solved, the part should be ready for manufacturing.

Chapter 5: Formability Analysis Tools

FTI CATSTAMP enables you to access new formability analysis tools. These include:

Forming Analysis icon ()

Blank Development icon ()

Springback Analysis icon ()

To access formability analysis tools

1. On the Main Menu, click **View** and then **Toolbars**.
2. Ensure that **Formability Analysis Tools** is enabled with a checkmark.

You can now access Formability Analysis icons on the CATIA toolbar.

Forming Analysis

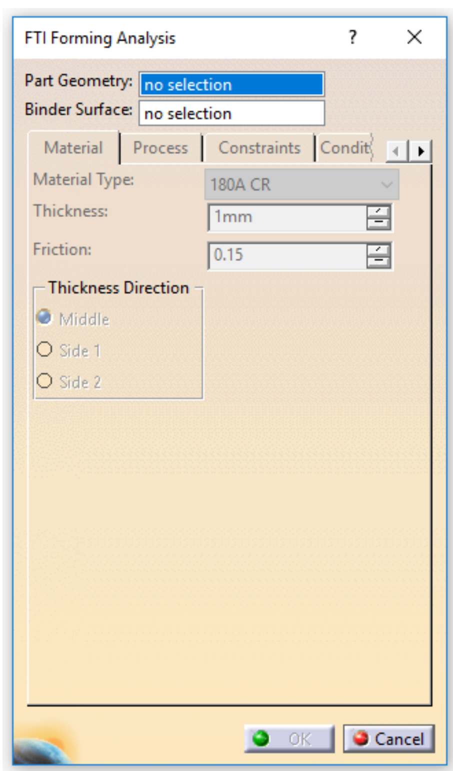
The **FTI Forming Analysis** dialog enables you to configure material properties, process, constraints, forming conditions, as well as other parameters. You can configure each tab independently, entering parameters, or leaving each blank.

Important It is recommended that you configure all required tabs before clicking **OK**. Clicking **OK** solves the forming analysis based on your input parameters, and creates a part geometry mesh that is added to the feature tree and named **FTICATSTAMP**.

This enables you to view formability results for the mesh. Click here to learn about viewing formability results.

To access the FTI Forming Analysis dialog

1. On the Formability Analysis Tool menu, click the **Forming Analysis** icon ().



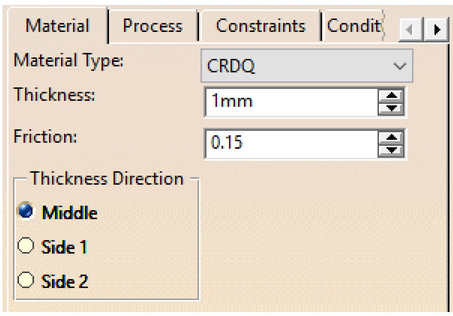
2. Click on your Part Geometry to select a part and enable the forming analysis tabs.

The Material Tab

The **Material** tab enables you to select or modify material and thickness parameters for your part geometry.

To configure the material tab

1. On the FTI Forming Analysis dialog, click the **Material** tab.



2. Next to **Material Type**, select a material from the drop-down menu.
3. Enter a **Thickness** in **mm** or use the arrows to increase or decrease the material thickness.
4. Enter a **Friction** value.

   Note Friction is applied to areas where the material is being drawn through a radius in a draw die, and is also used by forming conditions such as a Pressure Pad. This value defines the friction between the tooling and the blank material.
5. Under **Thickness Direction**, select from the following:

   Middle: Split the thickness and assume the value of the material thickness exists on both sides.

   Side 1: Choose the side that the material represents, and the direction of the material.

   Side 2

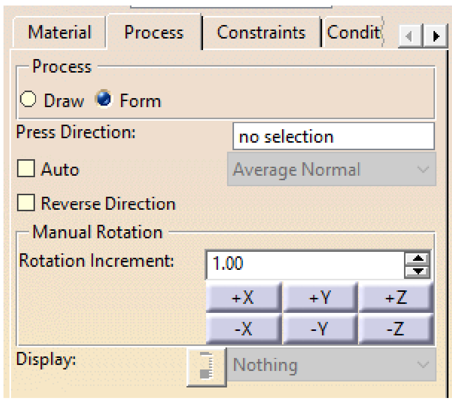
   > **Note** Thickness direction refers to the preferential grain direction of a material. This gives each material unique properties related to r-value and other material properties.
6. Click **OK** to close.

The Process Tab

The **Process** tab enables you to set the press direction as well as select a forming process.

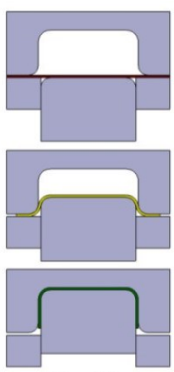
To configure process

1. On the FTI Forming Analysis dialog, click the **Process** tab.
2. In the field next to **Press Direction**, click in the graphics display area to select a press direction.



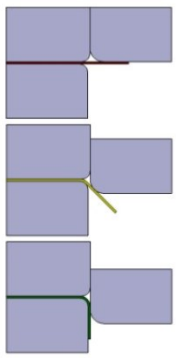
3. Under **Process**, select from the following:

Draw: A draw die is a specific form of die that wraps a flat sheet of metal around the punch while holding the workpiece around the cavity to control metal flow.



Draw Die

Form: A form Die is typically used in progressive dies and simple flanging operations. In this type of operation, the blank material is not pulled through a radius; it is simply bent around it. As a result, there are no additional strains produced by friction.


Form Die

4. Under **Press Direction**, select from the following:
   - **Auto**: CATIA automatically calculates the press direction. Select from the following methods:
     - **Average Normal**: The vector generated will be an average of the normal to all surfaces.
     - **Minimize Undercut**: The vector generated considers all angles on the part that could cause undercut and generates a vector that is less than 180 degrees to as many surfaces as possible.
     - **Minimize Depth**: The vector generated will minimize the draw depth of all surfaces.
   - **Reverse Direction**: Click to reverse the press direction.
5. Under **Manual Rotation**, click the X, Y, and Z icons to rotate the press direction. You can also enter a **Rotational Increment**.
6. Under **Display**, select the following:
   - **Undercut**: Displays areas that cannot be formed with an initial punch operation. These areas include flanges that must be created by a cam after the initial punch operation that curve under the initial stroke.
   - **Draft Angle**: Displays areas of the geometry set that allow for smooth ejection from the die.
   - **Draw Depth**

The Constraints Tab

The **Constraints** tab enables you to constrain elements of your part geometry. You can select several different elements such as:

Edges: Any element that is a border to a face or surface.

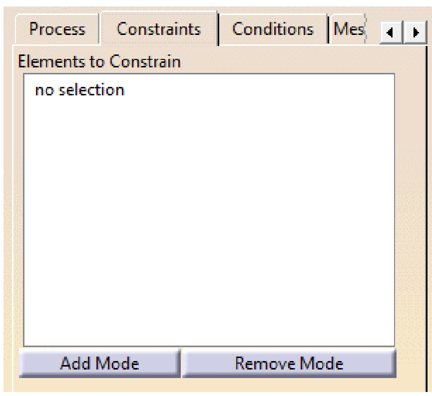
Faces: Any surface of the part geometry.

Lines: Any line that has been added to the part geometry to serve as a trim line or draw bead line.

Points: Any vertex on the part geometry.

To configure constraints

1. On the FTI Forming Analysis dialog, click the **Constraints** tab.



2. Under **Elements to Constrain**, click **no selection**.
3. Click on the geometry set to select an element.

   Note To remove a selected element, click **Remove Mode** and then click the element to remove it from the list.

4. Click **OK** to apply the forming constraints.

The Conditions Tab

The **Conditions** tab enables you to apply forming conditions to your part geometry. You can apply forming conditions to:

Edges: Any element that is a border to a face or surface.

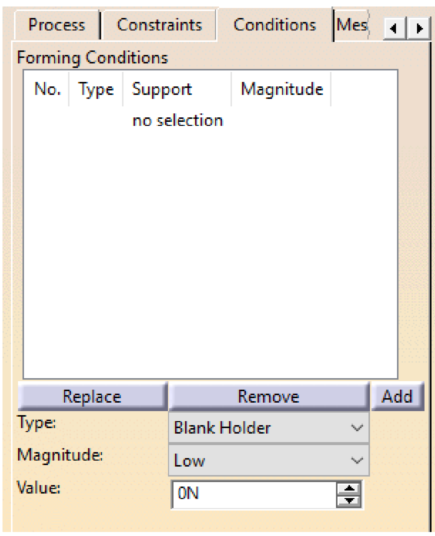
Faces: Any surface of the part geometry.

Lines: Any line that has been added to the part geometry to serve as a trim line or draw bead line.

Points: Any vertex on the part geometry.

To configure forming conditions

1. On the FTI Forming Analysis dialog, click the **Conditions** tab.



2. Select a forming condition **Type** drop-down menu:

   Blankholder: The blankholder is the upper and lower holding surfaces which control metal flow around a shape to be formed in a draw operation.

   Tension: Tension is the internal force or forces within a body. These forces can cause the extension or stretching of a material

   Draw Beads: A draw bead is a ridge constructed around a portion of a die cavity to control metal flow. A groove is added to the mating blank holder to allow clearance for the draw bead and die closing.

   Pressure Pad: The pressure bad controls the movement of material in the local area, and is simulated by restraining the movement of the mesh elements in the area under the pad.

3. Select a level of **Magnitude** from the drop-down menu:

   Low

   Medium

High

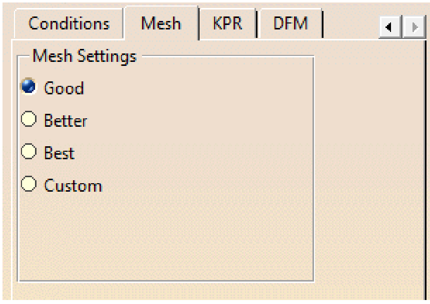
4. Enter a force **Value**, or use the arrows to increase or decrease the force value.
5. Click **Add** to add the forming condition to the **Forming Conditions** list.

The Mesh Tab

The **Mesh** tab enables you to select mesh generation settings for your part geometry.

To configure mesh settings

1. On the FTI Forming Analysis dialog, click the **Mesh** tab.



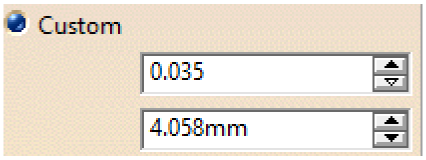
2. Under **Mesh Settings**, select from the following:

   Good

   Better

   Best

   Custom: Selecting custom enables you to manually enter Chord Deviation and Maximum Element Size for the mesh.



3. Click **OK** to save settings.

The KPR Tab

The **Key Product Requirements (KPR)** tab enables you to configure and apply a target stains to your geometry set. The KPR tab configures the following parameters:

Target Point: This is the point where the designated target strains are to be achieved. The point is selected by left clicking on the part surface.

Contact Point: This is the point where the part first contacts the binder wrap surface during the forming process. The point is selected by left clicking on the part surface.

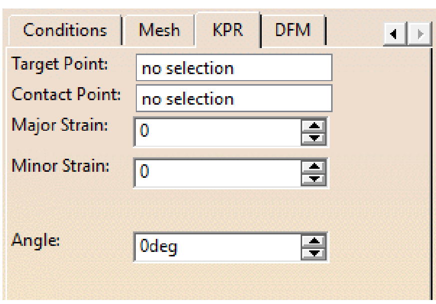
Major and Minor Strains: The target strain values to be achieved. The strain values must be greater than or equal to zero.

Angle: The orientation of the major strain axis relative to the part geometry is shown by the arrow located at the selected Contact Point.

Note If you configure KPR parameters, you cannot configure DFM parameters.

To configure KPR

1. On the FTI Forming Analysis dialog, click the **KPR** tab.



2. Click the field next to **Target Point**, then click to select a target point on your geometry set.
3. Click the field next to **Contact Point**, then click to select a contact point on your geometry set.
4. Enter a **Major Strain** and **Minor Strain** value, or use the arrows to increase or decrease the value.
5. Enter an **Angle** for the target strain in degrees, or use the arrows to increase or decrease the value.
6. Click **OK** to save and close.

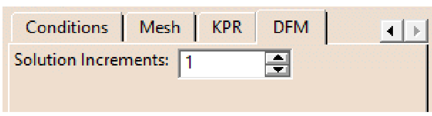
The DFM Tab

The **Design for Manufacture (DFM**) tab enables you to manually configure solution increments for Auto Blankholder Force. The **blankholder** is the upper and lower holding surfaces which control metal flow around a shape to be formed in a draw operation. Auto Blankholder Force runs multiple analyses with different blankholder forces to produce various outputs and find a good compromise between thinning and wrinkling.

Note If you increase solution increments to any value greater than 1, the KPR tab is disabled.

To configure the DFM tab

1. On the FTI Forming Analysis dialog, click the **DFM** tab.



2. Enter a **Solution Increments** value. Solution Increments set the number of solutions provided in post processing.
3. Click **OK** to save and close.

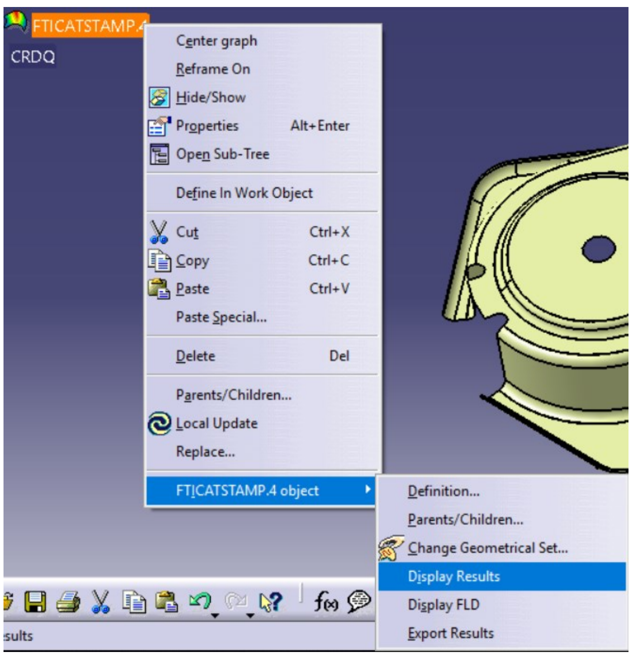
Viewing Formability Results

Once you have configured and applied all Formability Analysis tools and created a **FTICASTAMP** branch in the feature tree, you can view formability results such as Safety Zones, or Forming Limit Diagrams (FLD) for the geometry set.

Note Each time that you conduct a Formability Analysis on a Geometrical Set, a new branch named FTI CATSTAMP is added to the Feature Tree. This enables you to view formability results for multiple forming processes.

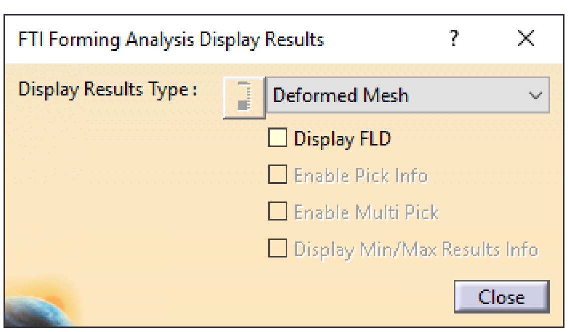
To view formability results

1. On the Feature Tree, right-click on **FTICATSTAMP** and then select **FTICATSTAMP object** > **Display Results**.



2. In the **Display Results Type** drop-down menu, click to select a formability result.

   Note Depending on the results type selected, you can enable **Smooth Display** to smooth contours of the results plot.



The following results are available for your review:

- **Deformed Mesh**: Displays deformations in the mesh of the part after forming.
- **Safety Zones**: Demonstrates if a part is formable by displaying qualitative color areas ranging from Safe to Split.
- **Forming Zones**: Describes the state of stretch in the part. It is a qualitative plot and each color represents a different zone. Each zone produces a different panel quality and characteristic. The forming zones can be used to identify the strain state of the panel.
- **Safety Margins**: Displays the relative strain between the Forming Limit Curve (FLC) and the major strain value at each node.
- **Thickness Strain**: Displays the change in thickness of material due to forming. A negative value indicates that the part is thinning out. Severe thinning may indicate splitting and thickening may indicate wrinkling (especially in thin materials).
- **Thickness**: Displays the change in thickness of the material due to forming, shown as thinning and thickening in millimeters (mm).
- **Major Strain**: Displays the largest principal strain in the sheet surface. It is often measured from the major axis of the eclipse resulting from the deformation of a circular grid.
- **Minor Strain**: Displays the principle strain in the sheet surface in the direction perpendicular to the major strain.
- **Equivalent Strain**: Displays the hypothetical strain representing the sum of the Thickness, Major, and Minor strain values in a uniaxial tension test.
- **Equivalent Stress**: Displays the hypothetical stress required to deform the material by the equivalent strain amount in a uniaxial tension test.
- **Displacements**: Displays displacements between the part and blank.
- **Displacements 2D**: Displays the node displacement along the XY plane between the part and blank.
- **Displacements X**: Displays displacements in the X direction only.
- **Displacements Y**: Displays displacements in the Y direction only.
- **Displacements Z**: Displays displacements in the Z direction only.

Depending on the results you choose to display, you can click to enable and open **Display FLD**, which will open a separate Forming Limit Diagram window. You also can access the following options:

- **Enable Pick Info**: Click a point on the geometrical set to show specific display results for that point.
- **Enable Multi Pick**: Click multiple points on the geometrical set to show display results for that point.

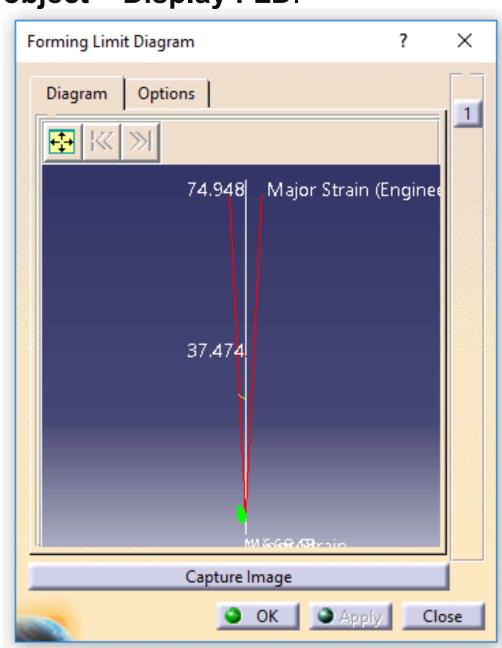
- **Display Min/Max Results Info**: Display the Minimum and Maximum values on the geometrical set, based on your initial pick info point.

Viewing the Forming Limit Diagram (FLD)

In addition to viewing Formability plots, you can also view a **Forming Limit Diagram (FLD)** based on your formability analysis. Formability describes whether the part is safe or will have areas with a tendency to split or wrinkle. The FLD is a map of the strain state of a model. The strains of each node are plotted onto the major and minor strain axes. The position of each point on the FLD can be used to determine what forming mode and the formability of the part in a local area.

To view the FLD

1. On the Feature Tree, right-click on **FTICATSTAMP** and then select **FTICATSTAMP object** > **Display FLD**.



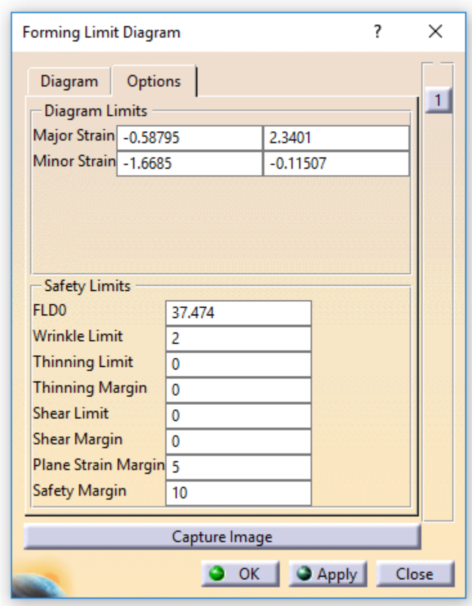
2. Click the **Center** icon (icon) to reposition the FLD plot to the center axis.
3. Click **Capture Image** to save the FLD plot as a **JPEG (.jpg)** file.
4. Click **OK** to close.

Options Tab

You can also configure parameters for your FLD including Diagram Limits and Safety Limits.

To configure FLD options

1. On the **Forming Limit Diagram** dialog, click the **Options** Tab.



2. Under **Diagram Limits**, configure the following:

   Major Strain: Enter the lowest and highest limits for the largest principal strain.

   Minor Strain: Enter the lowest and highest limits to the principle strain in the sheet surface in the direction perpendicular to the major strain.

3. Under **Safety Limits**, configure the following:

   FLD0: Enter the maximum amount of strain allowed one direction when there is no strain in the transverse direction. It is shown as the intersection of the Forming Limit Curve (FLC) and the major strain axis on a Forming Limit Diagram (FLD).

   Wrinkle Limit: Enter a value to limit the area appearing to the left of the Major Strain line on the FLD.

   Thinning Limit: Enter a value to limit the thinning line.

   Thinning Margin: Enter a value to change the position of the yellow line on the lower left side of the plot.

   Shear Limit: Enter a custom offset for marginal splitting zones.

   Shear Margin: Enter a value to change the position of the yellow line on the top left side of the plot.

Plane Strain Margin: Enter a custom value to change the minor strain percentage of the major strain and define the plane strain zone.

Safety Margin: Enter a value to define the marginal splitting zones.

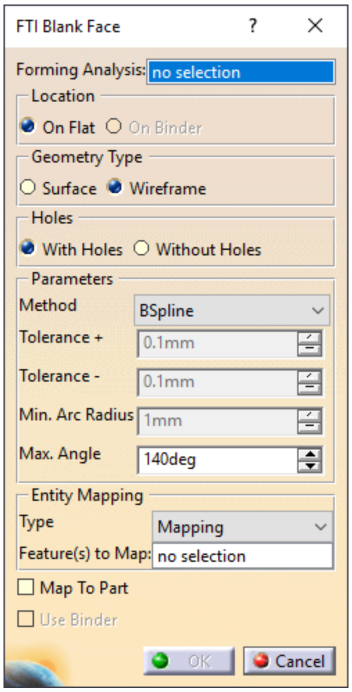
4. Click **Apply** to save and **OK** to close.

Blank Development

Once you have configured and run Formability analysis, you can use Blank Development to create and configure a blank.

To create and configure a blank

1. On the Formability Analysis Tool menu, click the **Blank Development** icon ().



2. On the Feature Tree, click to select **FTICATSAMP**. This enters the information in the **Forming Analysis** field.
3. Under **Location**, select from the following:

   On Flat: Creates a flat blank on the plane perpendicular to the tipping direction.

   On Binder: Creates a blank outline projected onto the binder surface.

   Note Setting the location to On Binder means that only outline is available and not surface.
4. Under **Geometry Type**, select from the following:

   Surface: Creates the blank as a surface.

   Wireframe: Creates the blank as an outline.
5. Under **Holes**, select from the following:

   With Holes: Creates a blank to include any unfilled holes.

   Without Holes: Creates a blank but removes any unfilled holes.

6. Under **Parameters**, configure the following:

   Method: Click the drop-down menu to select:

   - **BSpline**: Creates the blank boundary as a bspline with no tolerance, and uses the nodes on the boundary to create the blank.
   - **Line**: Creates the blank boundary using only lines.
   - **Lines and Arcs**: Creates the blank boundary by fitting lines and arcs based on the specified tolerance.

   Tolerance + : Configure the maximum tolerances for lines, or lines and arcs.

   Tolerance – : Configure the minimum tolerance for lines, or lines and arcs.

   Min. Arc Radius: Enter minimum arc radius size before a corner is created instead of an arc.

   Max. Angle: The maximum arc angle determines at what point a line will be created instead of an arc.

Under **Entity Mapping**:

   Type: Select from the following:

   - **Mapping**: Map features of the geometry such as points or edges onto the blank.
   - **Part Weight**: Calculate the weight of a specific part. This is useful when trying to understand the differences in part weights between parts with or without holes for example.

   Features To Map: Click no selection and then click the geometry set to select a feature to map.

7. Click to enable **Map To Part**.
8. If you are using a Binder, you can click to enable **Use Binder**.
9. Click **OK** to save and close.

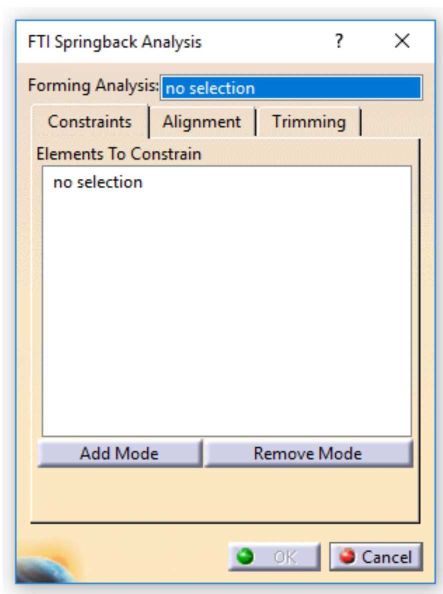
Springback Analysis

Springback analysis enables you to select a completed forming analysis and display springback in the material. Springback, also known as *Elastic Recovery*, occurs when forces of forming are released and the blank "springs back" from elastic deformation. This is important to understand because it can affect the final dimensions of the part.

A blank undergoes plastic and elastic deformation during forming. **Plastic** deformation occurs when there is a permanent stretch in the blank material as it is formed. **Elastic** deformation is a deformation that is not permanent.

To access the FTI Springback Analysis dialog

1. On the Formability Analysis Tool menu, click the **Springback Analysis** icon ().



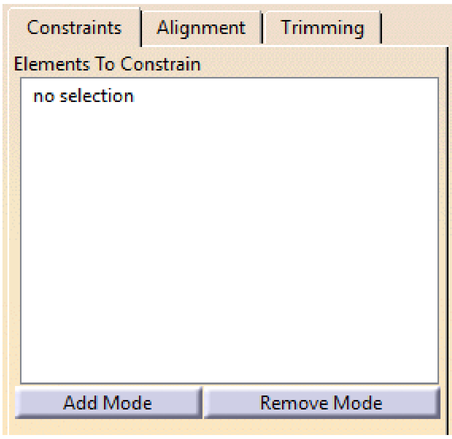
2. On the Feature Tree, click to select **FTICATSAMP**. This enters the information in the **Forming Analysis** field.

The Constraints Tab

The **Constraints** tab enables you to select elements on your geometry set and constrain them to reduce springback.

To configure springback constraints

1. On the FTI Springback Analysis dialog, click the **Constraints** tab.



2. Under **Elements to Constrain**, click **no selection**.
3. Click on the geometry set to select an element.

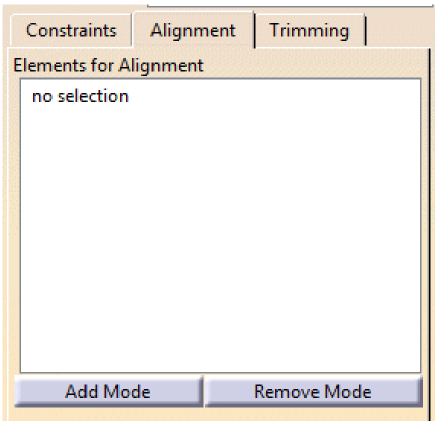
   Note To remove a selected element, click **Remove Mode** and then click the element to remove it from the list.
4. Click **OK** to apply the springback constraints.

The Alignment Tab

The **Alignment** tab enables you to select elements to align to on your geometry set.

To configure alignment

1. On the FTI Springback Analysis dialog, click the **Alignment** tab.



2. Under **Elements to Constrain**, click **no selection**.
3. Click on the geometry set to select an element.

   Note To remove a selected element, click **Remove Mode** and then click the element to remove it from the list.

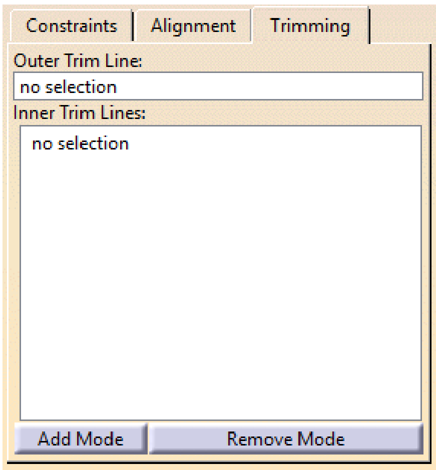
4. Click **OK** to align to the selected elements.

The Trimming Tab

The **Trimming** tab enables you to select an Outer Trim Line and multiple Inner Trim Lines to trim.

To configure trimming

1. On the FTI Springback Analysis dialog, click the **Trimming** tab.



2. Under **Outer Trim Line**, click **no selection**, and then click on the geometry set to select an outer trim line.
3. Under **Inner Trim Lines**, click **no selection**, and then click on the geometry set to select one or more inner trim lines.

   Note To remove a selected element, click **Remove Mode** and then click the element to remove it from the list.

4. Click **OK** to select trim lines and close.

Chapter 6: Springback

In this section:

- Springback
- Springback Analysis
- Process Conditions
- Springback Alignment
- Springback Fixity
- Springback Trimming
- Forming Displacement
- Reflect Lines

Springback

When sheet metal is formed, it undergoes two types of deformation: plastic and elastic. The plastic deformation permanently deforms the material. When the part is removed from the die and the forming forces are taken away, the elastic deformation will be unloaded and the part will experience springback.

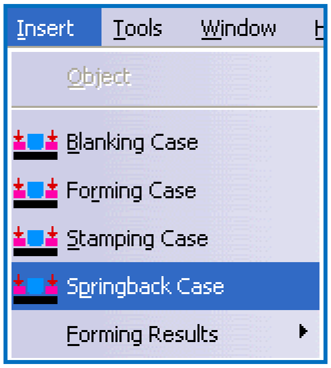
The type of material used in forming has a significant impact on springback. Higher stresses are required to form stronger materials which will produce more elastic recovery. Different materials (e.g. aluminum) have different elastic modulus values that will also change the springback.

Although it is often neglected, springback can have an important effect on the resulting part shape. A large amount of springback will make it difficult to control the final dimensions of the part.

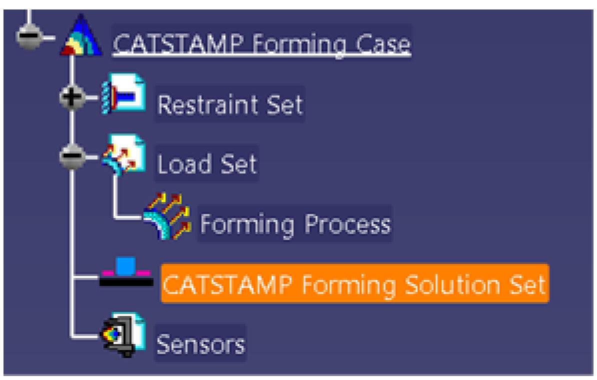
The springback solver determines the shape of a formed part after it has been released from the tool and the elastic recovery of the part (springback) has occurred.

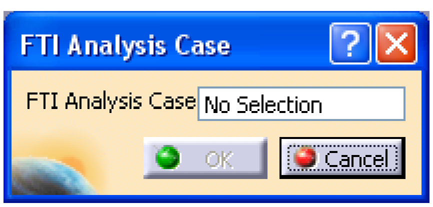
Springback Analysis

To analyze a part's springback in CATSTAMP, select **Springback Case** from the **Insert** menu.



You will be prompted to select an analysis case in the **FTI Analysis Case** dialog box. Select the appropriate Forming Solution Set from the feature tree. The springback results will be calculated based on the forming analysis results from the selected solution set.





While the **CATSPRINGBACK Case** is activated, the **CATSTAMP Forming Solution Set** will be deactivated until the **CATSTAMP Forming Solution Set** is selected once again as the current case.


CATSTAMP Forming Case
Restraint Set
Load Set
CATSTAMP Forming Solution Set
Sensors
CATSPRINGBACK Case

Process Conditions

To input various process conditions open the **Process Condition** toolbar on the **Forming Toolbar** and select from the following choices:

Springback Alignment aligns the deformed mesh results with reference geometry.

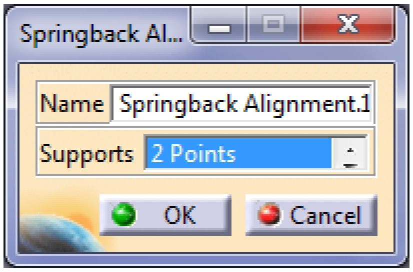
Springback Fixity simulates a clamped condition around the outer boundary of the part.

Springback Trimming removes surface faces from the CATSTAMP analysis geometry before running springback analysis.

Clamp simulates a clamped condition for specified CAD geometry. Refer to section 3.12 on pg. 23 for more information.

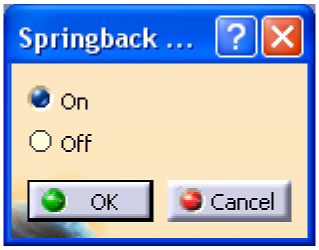
Springback Alignment

To apply **Springback Alignment** to the part geometry open the **Process Condition** toolbar and select **Springback Alignment**. For best results, create points on the original part geometry at the desired alignment locations. In the alignment dialog, select the desired points. This will align the part with the corresponding locations on the deformed mesh using a best-fit approach.



Springback Fixity

The **Springback Fixity** tool is used to analyze parts that will be secured to another panel around their entire boundary; such as hemmed outer skin panels. Open the **Process Conditions** toolbar and select **Springback Fixity** to turn on the entire outer boundary fixity during springback. This will constrain the springback solution such that the boundary of the part does not deflect. This function is very useful for identifying the potential for surface distortion on visible panels.



Springback Trimming

The springback exhibited by a part after forming will change when material (such as large holes or addendum) is trimmed off **Springback Trimming** is used to correctly simulate this scenario. Open the **Process Conditions** toolbar and select **Springback Trimming**. Select an object from the feature tree that includes all of the surfaces to be kept after trimming. Note that this object must be a parent of the external view.

Springback Results Plots

Displays the various displacement components as calculated between the part and blank.

Direction	Description
Deformed Mesh	This result will display the mesh of part after springback. It provides a convenient way of overlaying the springback result with the original part.
3D	The total node displacement between the designed part and the part experiencing springback. The square root of the X displacement squared plus the Y displacement squared plus the Z displacement squared.
2D	Node displacement between the designed part and the part experiencing springback in the X-Y plane. The square root of the X displacement squared plus the Y displacement squared.
X	Node displacement in the X direction between the designed part and the part experiencing springback.
Y	Node displacement in the Y direction between the designed part and the part experiencing springback.
Z	Node displacement in the Z direction between the designed part and the part experiencing springback.

Notes:

- You can use the animate button on the Analysis Tools toolbar to animate the deformed mesh plot.
- The Deformation Scale Factor tool on the same toolbar can be used to magnify the springback effect for visibility.
- When showing the deformed mesh plot, it is helpful to show the part geometry as a basis for comparison.

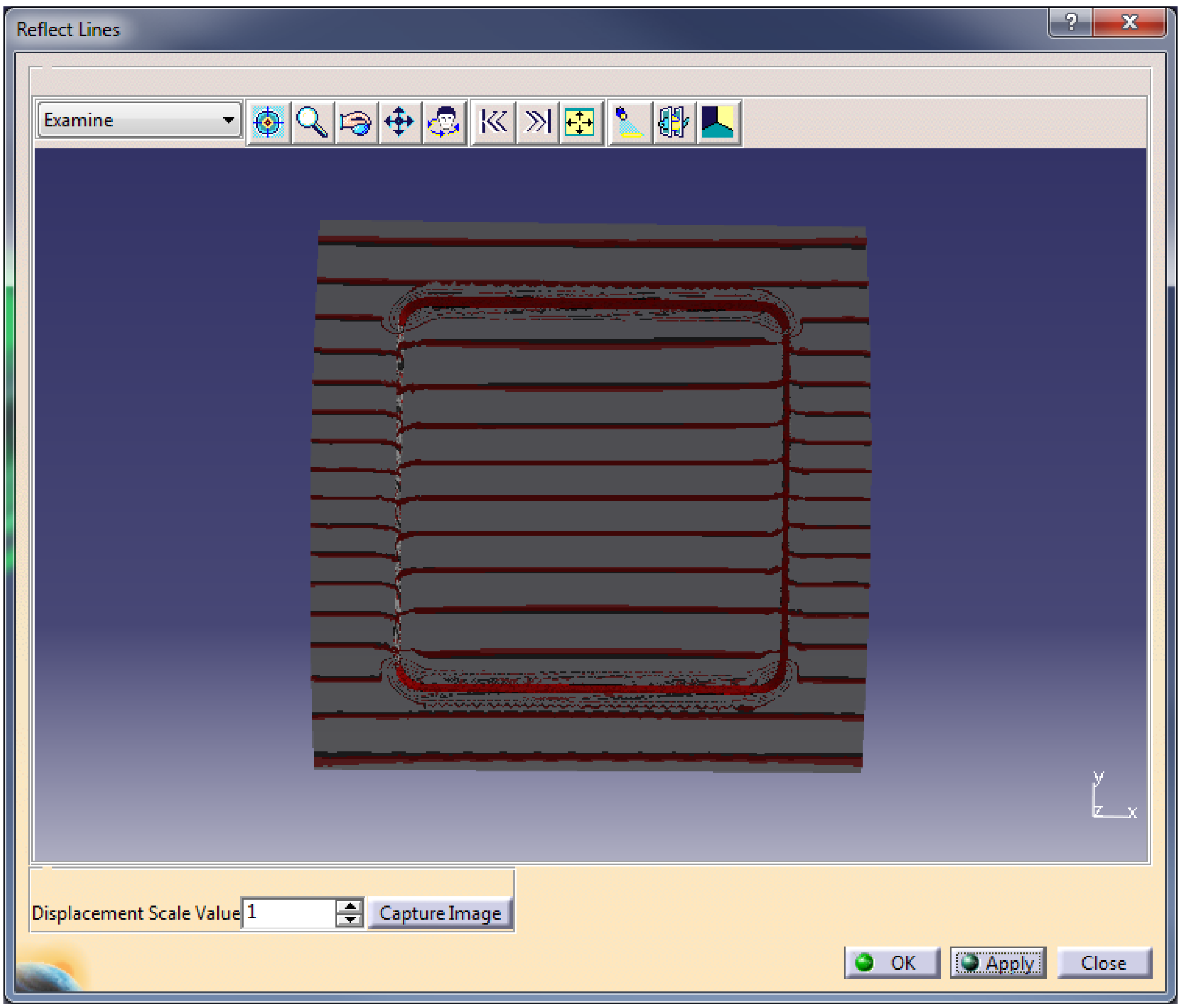
Reflect lines

Reflect Lines are an important way of measuring Class 'A' panel surface quality. They are a direct measurement of what the customer will see when they view the completed vehicle.

In practice, **Reflect Lines** are the reflection of a straight light source off the curved three dimensional surface of the panel.

The Reflect Line viewer shows a series of reflect lines on the panel. The blank lines represent the reflect lines based on the original part geometry (before springback) and the red lines represent the reflect lines on the geometry after springback. Areas where these two lines diverge indicates areas of high distortion.

Select the **Reflect Line** icon on the **Forming Toolbar** to open the viewer as a separate window.



As with physical panels, the reflect lines should be viewed from multiple vantage points. This highlights each part feature with the light source angle that best displays the surface quality. Different light source locations can be shown in the viewer by rotating the part to the desired angle and then clicking the **Apply** button. The part can then be rotated to examine the reflect lines of any part area. The light source will not be changed until the apply button is selected.

The relative scale between the ideal reflect lines (shown in black) and the stamped part reflect lines (shown in red) can be adjusted with the displacement scale value option. Changing the scale can highlight the subtler deviations for easier analysis.

Notes:

- To properly display the results in the **Reflect Line** viewer, the High (Alpha Blending) option must be selected for the **Transparency Quality** setting in Tools->Options->Display->Performance.
- The Displacement Scale Value will increase magnitude of the difference in the springback lines versus the theoretical designs for easier spotting of problem areas.
- The reflect lines are created on the finite element mesh. In areas of low mesh density, this may result in jagged reflect lines. Refining the mesh will improve the quality of the reflect line display in such cases.

Export

While working on a Springback case, the Export icon will allow you to export the springback data in a variety of formats:

- **NASTRAN (.nas or .dat)**: exporting in NASTRAN format will produce a file containing the “sprung” mesh (after springback)
- **LS-Dyna (.dyn)**: this option will export an LS-Dyna model file containing the sprung mesh
- **Text (.txt)**: this option will export a text file containing the nodal displacements occurring during springback. This file can then be used by the CATIA RSO module for morphing the geometry to compensate for springback.
- **STL File (.stl)**

Chapter 6: Nesting Layout

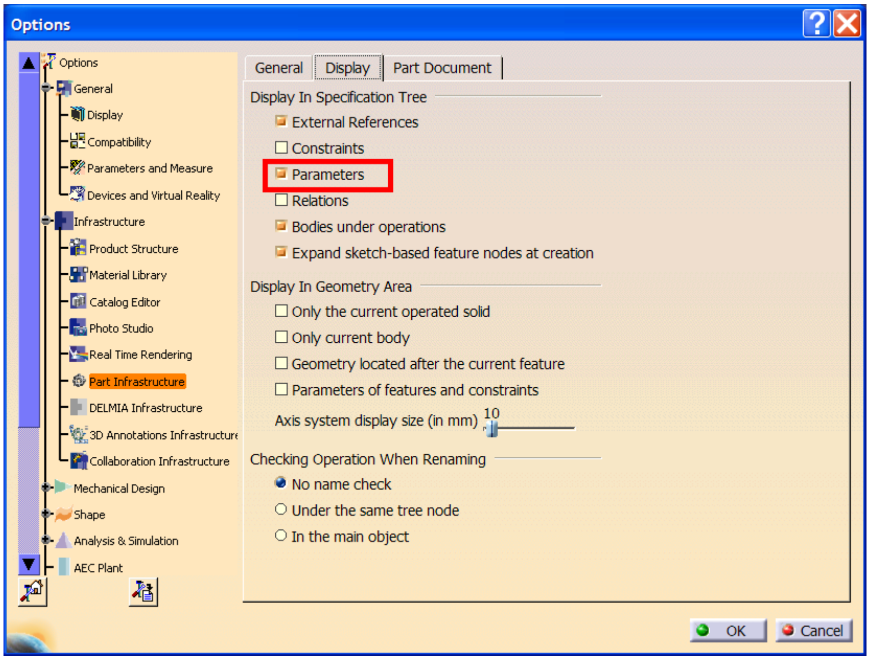
In this section:

- Displaying Parameters
- Defining Cost Parameters
- Defining Material
- Nesting Parameters
- Viewing Nesting Layout Results
- Viewing Other Nesting Layout Results
- Layout Editing
- Associative and Regenerative Layout

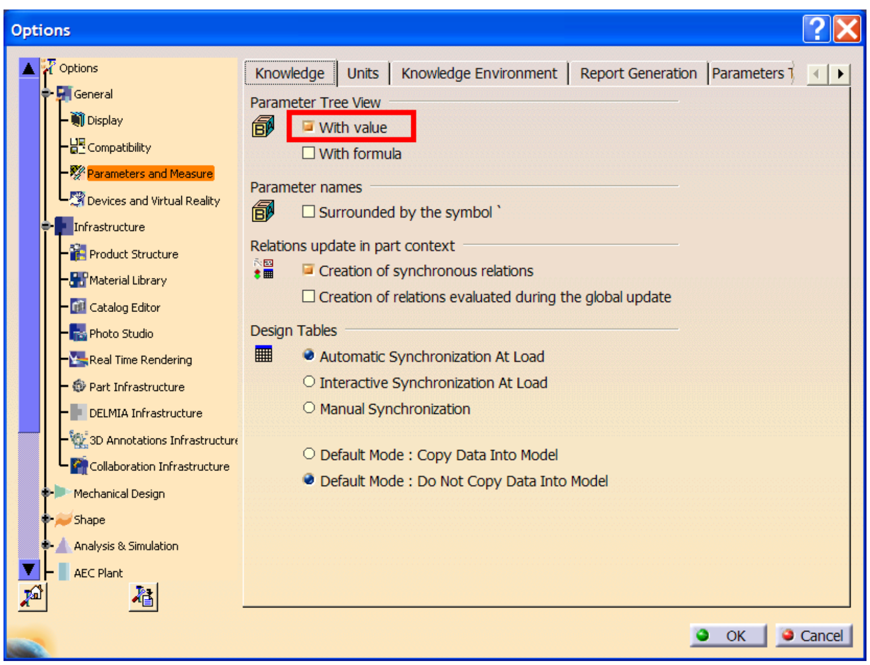
Displaying Parameters

CATNEST uses parameters to input layout and cost parameters. In the **Options** window located under the **Tools** menu, the following options must be set to properly utilize CATNEST.

Turn on the display of **Parameters** in the Specification Tree.

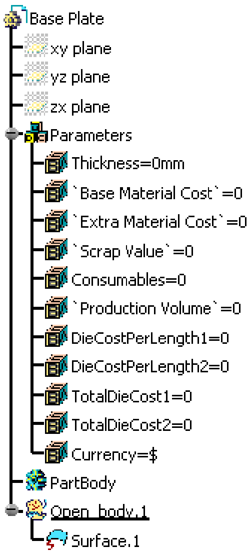


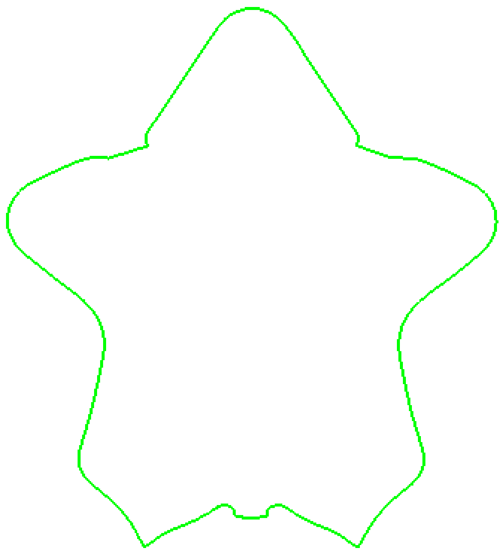
Turn on the display of the Parameters **Value** in the Specification Tree.



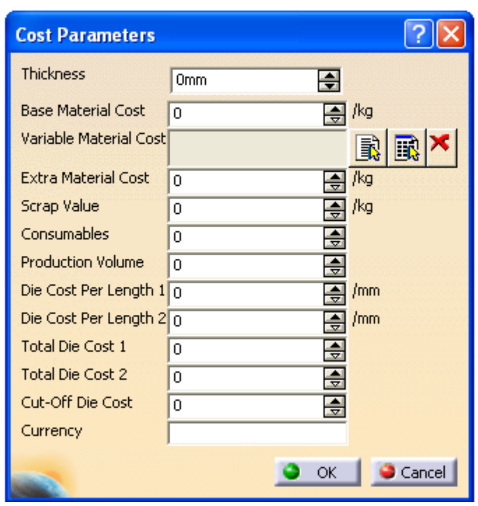
Defining Cost Parameters

To set up the nesting calculation, some parameters need to be defined. These are entered under the Parameters branch of the Specifications Tree.





You can specify cost parameters by clicking the Cost Parameters icon on Nesting toolbar and key in the value in the Cost Parameters dialog.



NOTE: By default, the Thickness cost parameter is read from the part geometry. If part thickness is defined using thin parts attribute, shell entity or thick surface, the thickness value will show up automatically. This parameter is not linked to the analysis workbench directly, so the analysis thickness will not used by the cost parameters.

Thickness: part model material thickness

Base Material Cost: fixed cost of the material per weight unit

Variable Material Cost: cost specified by coil thickness and width

Extra Material Cost: any other material costs measured per unit weight of material. e.g., lubricants, coatings, handling, etc.

Scrap Cost: the value of the scrap material

Consumables: any additional costs that cannot be measured in $/kg. e.g., electricity, labor, etc.

Production Volume: the annual number of blanks that will be produced or the total number of blanks on a program basis.

DieCostPerLength1: represents the cost of the die based on the perimeter of the blank shape for one-up nesting layouts.

DieCostPerLength2: represents the cost of the die based on the perimeter of the blank shape for two-up nesting layouts.

TotalDieCost1: the overall value of the die expressed in the current currency for one-up nesting layouts.

TotalDieCost2: overall value of the die expressed in the current currency for two-up nesting layouts.

Cut-Off Die Cost: overall value of the cut-off die expressed in the current currency

Currency: the current currency

NOTE: The cost parameters can also be modified by double clicking on the parameter name located under the Parameters branch of the Specifications Tree.

Variable Material Cost

The additional costing parameters are imported as a table. The imported table specifies the material price per kilogram, depending on sheet thickness and coil width. If a table is imported the Base Cost is grayed out.

Creation of the table can be done using a predefined Excel template. This will ensure that exported ASCII data will always be consistent. The table should be exported in either .txt or .csv format.

Format of the Table

The entry in the first column/row should generally be defined as 0. It specifies the cost parameters for the range from 0 to the next value or from the value in the first column/row to the next value.

If the parameter appears for which the thickness/width range is not defined, then the resulting cost will be 0.

The specified price applies to the range between the specified value and the next specified value, i.e. 86 applies to thickness between >0.6 and <=0.7 and a coil width of <= 750 for the table shown below.

The value in the last column/row is taken for the thickness/width higher than the specified value, i.e. coil width is 2000mm and thickness is 0.75, base cost value is 72 for the table below. For 1mm thickness and 2000mm width, 71 is applied for the table below.

If the calculated width is less than the smallest width in the table then the cost will be zero by definition.

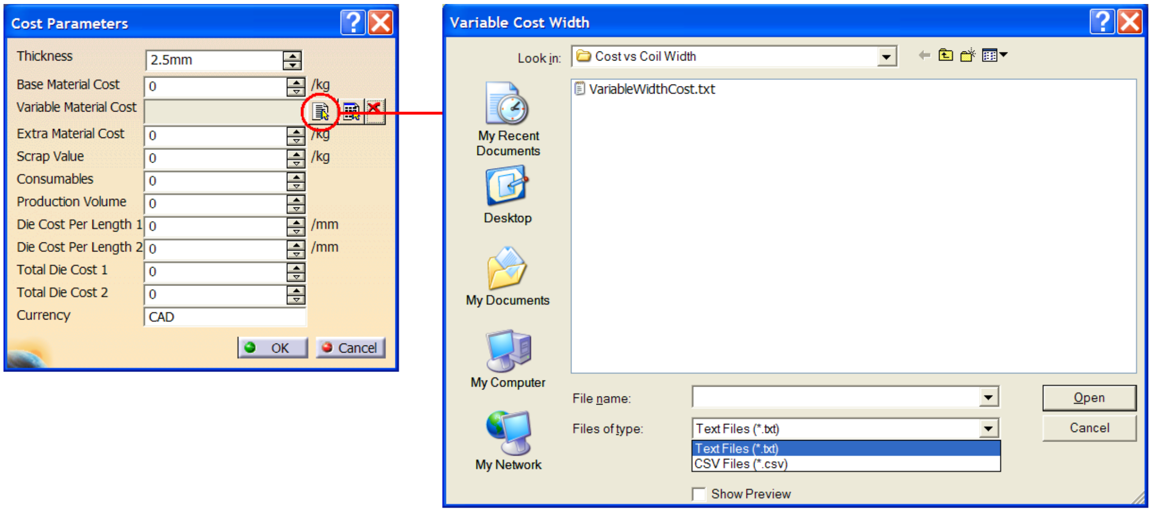
VariableWidthCost.txt - Notepad

	500	750	900	1100
0.5	87	83	78	74
0.6	86	82	77	73
0.7	85	81	76	72
0.8	84	79	75	71

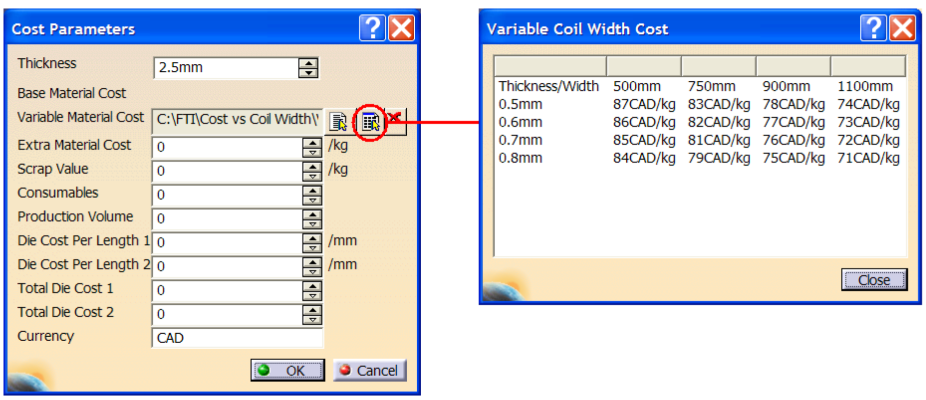
The thickness and width must be specified in mm. The cost value is specified as $/kg. The currency type is taken from the Cost Parameters. If no currency is defined, then USD will be used as the default.

Importing a Variable Material Cost Table

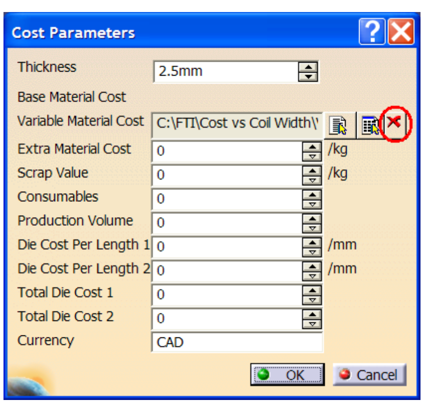
To import a variable material cost table open the Cost Parameters dialog, select the import icon, and select the text and csv file to be imported.



To view the table after import, select the view icon. Notice that the units have been added to the table and the Base Material Cost has been grayed out.



To remove the Variable Material Cost table, select the remove icon.

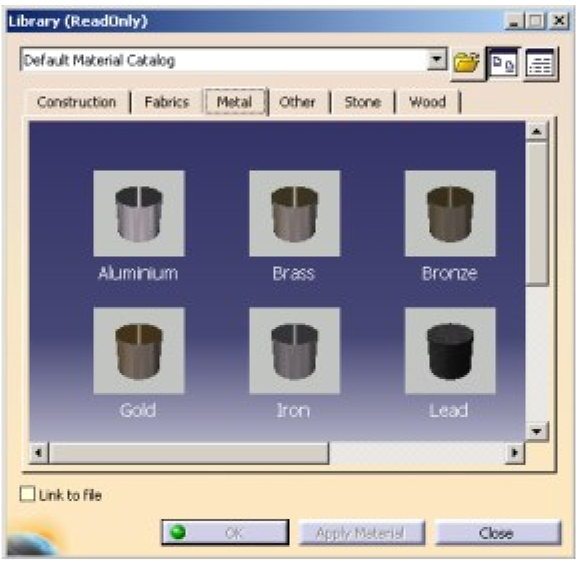


Defining Material

CATNEST uses the properties of the material assigned to calculate the gross weight of the part model.

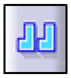
To assign a material to your model, select the **Apply Material** icon from the **Apply Material** toolbar.

The Library dialog below will appear. Select a material and select the geometry to which you want to apply the material. The easiest way to do this is to select the **Part1** item at the top of the feature tree. This will apply the material to the entire model.

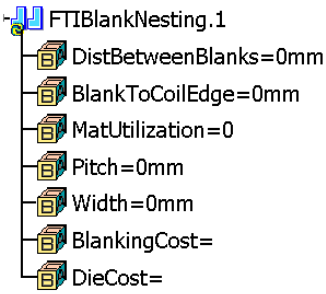


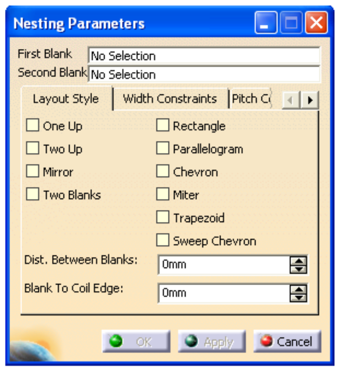
Nesting Parameters

To set up the nesting calculation, some parameters need to be defined. These are entered in the Nesting Parameters dialog box. To open the Nesting Parameters dialog box select the Compute Nesting Layout icon on the Nesting toolbar.



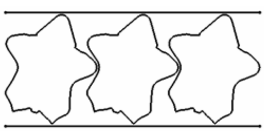
An **FTIBlankNesting.1** branch is added to the Specifications Tree and the Nesting Parameters dialog box opens when the Compute Nesting Layout icon is selected.



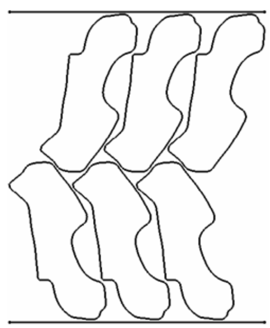


Select the blank surface (First Blank) to define the geometry to be used in the nesting layout. For a Two Blanks nesting layout a Second Blank must also be defined.

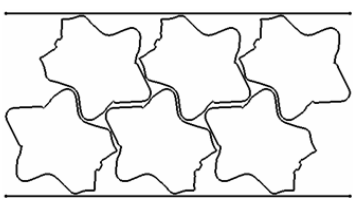
On the **Layout Style** tab of the Nesting Parameters dialog box specify the nesting layouts to be calculated.



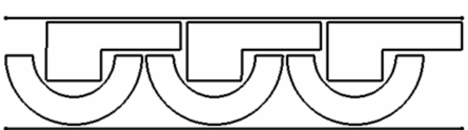
One Up



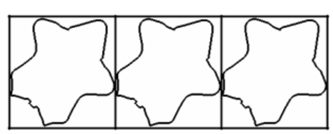
Mirror



Two Up



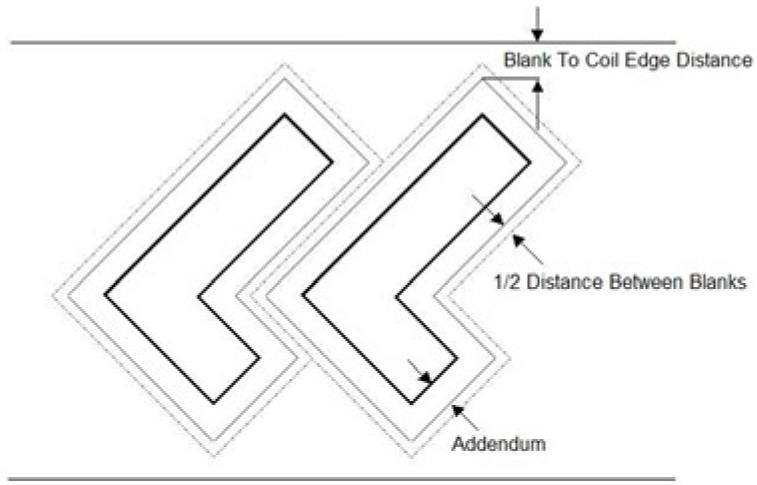
Two Blanks



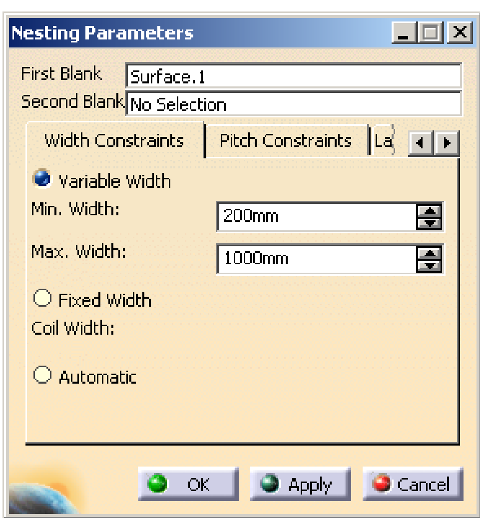
Cut-off Die

Layout constraints can be defined on the other tabs in the Nesting Parameters dialog box. Once all the nesting parameters have been defined select the OK button to start the nesting layout calculation.

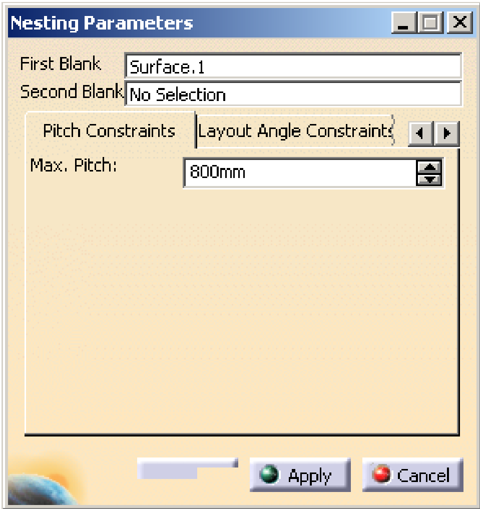
You can also specify the Dist. Between Blanks and Blank to Coil Edge parameters. The Dist. Between Blanks is the minimum distance between the nested blanks. The Blank to Coil Edge is the minimum distance between the nested blanks and the edge of the coil.



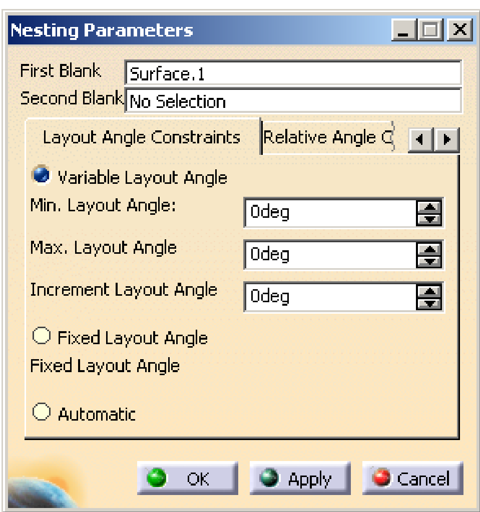
If the width of the coil is known prior to nesting, this information can be added as a constraint on the **Width Constraints** tab of the Nesting Parameters dialog box. If a Variable Width constraint is added, only the nesting layouts that fall within the minimum and maximum values will be calculated. If a Fixed Width is defined, only nesting layouts with the fixed width value will be calculated. Defining an Automatic constraint will calculate the coil width for the lowest cost. The maximum coil width is defined as 72 inches or 1828.8 mm. This value can be overwritten by keying in a different value in the Max. Width: field.



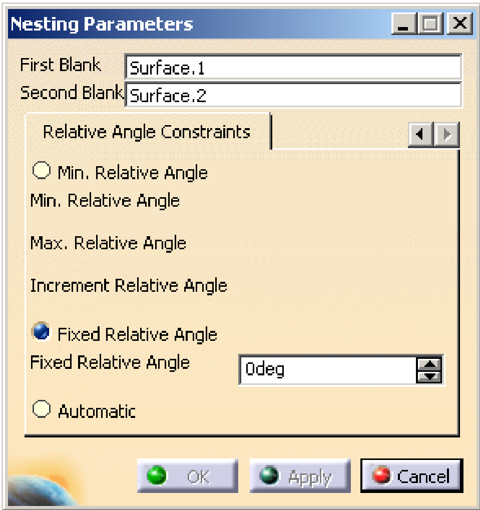
Pitch refers to the relative distance between the blank shapes. If the pitch of the nesting layout is known prior to nesting, this information can be added as a constraint on the **Pitch Constraints** tab of the Nesting Parameters dialog box. If a maximum pitch constraint is added, only the nesting layouts with a pitch equal to or less than the specified value will be calculated.

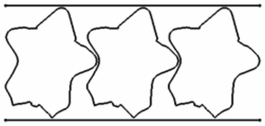


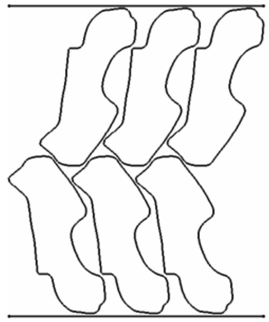
To control how the first blank is rotated on the coil an angle constraint can be defined on the **Layout Angle Constraints** tab of the Nesting Parameters dialog box. If a Variable Layout Angle constraint is added, only the nesting layouts that fall within the minimum and maximum values will be calculated. If a Fixed Layout Angle is defined, only nesting layouts with the fixed layout angle value will be calculated. Defining an Automatic constraint will calculate the layout angle for the lowest cost.

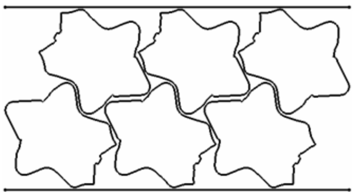


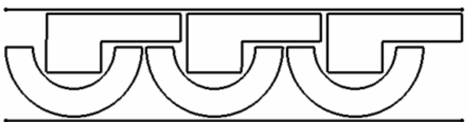
To control how the second blank is rotated on the coil an angle constraint can be defined on the **Relative Layout Angle Constraints** tab of the Nesting Parameters dialog box. If a Variable Relative Angle constraint is added, only the nesting layouts that fall within the minimum and maximum values will be calculated. If a Fixed Relative Angle is defined, only nesting layouts with the fixed relative angle value will be calculated. Defining an Automatic constraint will calculate the relative angle for the lowest cost. Defining a Relative Angle Constraint only is useful for Two Up, Mirror and/or Two Blank layouts.

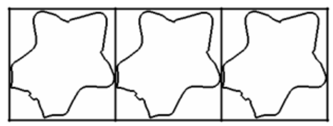



One Up


Mirror

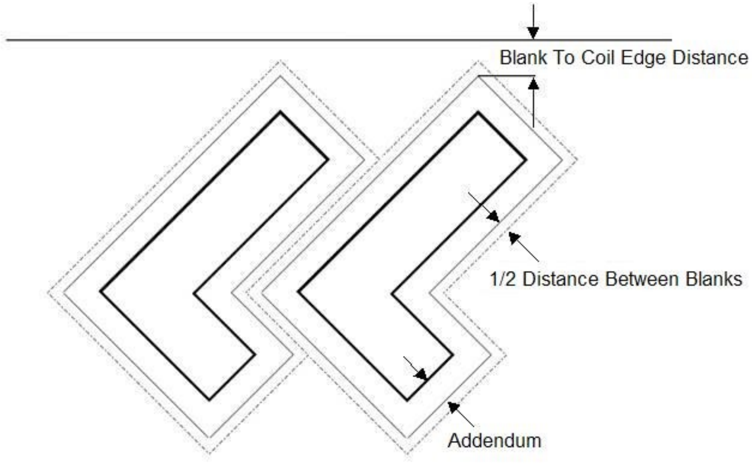

Two Up


Two Blanks

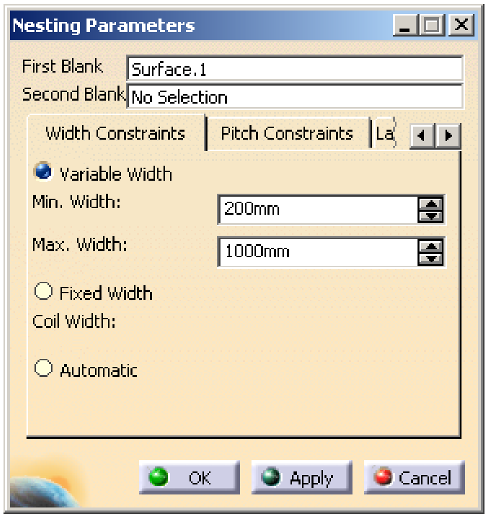

Cut-off Die

Layout constraints can be defined on the other tabs in the Nesting Parameters dialog box. Once all the nesting parameters have been defined select the OK button to start the nesting layout calculation.

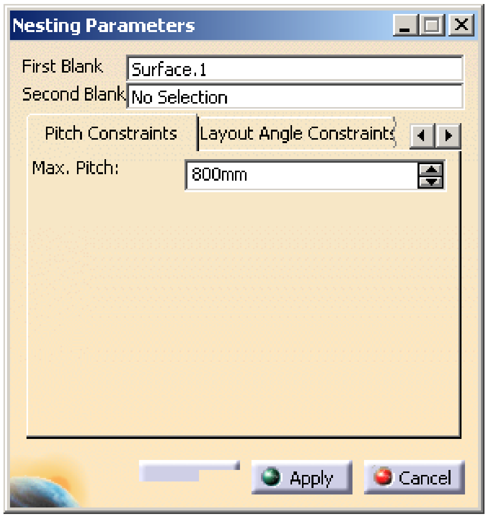
You can also specify the Dist. Between Blanks and Blank to Coil Edge parameters. The Dist. Between Blanks is the minimum distance between the nested blanks. The Blank to Coil Edge is the minimum distance between the nested blanks and the edge of the coil.



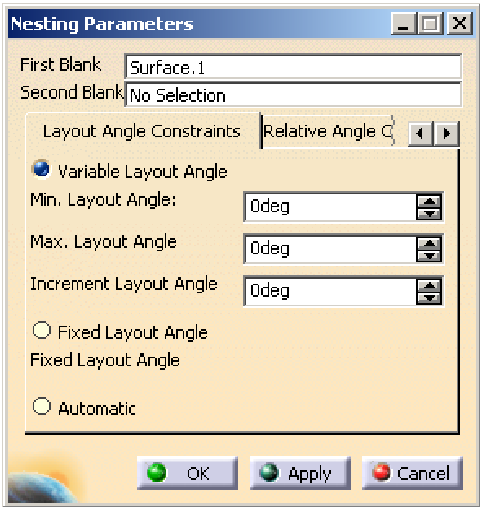
If the width of the coil is known prior to nesting, this information can be added as a constraint on the **Width Constraints** tab of the Nesting Parameters dialog box. If a Variable Width constraint is added, only the nesting layouts that fall within the minimum and maximum values will be calculated. If a Fixed Width is defined, only nesting layouts with the fixed width value will be calculated. Defining an Automatic constraint will calculate the coil width for the lowest cost. The maximum coil width is defined as 72 inches or 1828.8 mm. This value can be overwritten by keying in a different value in the Max. Width: field.



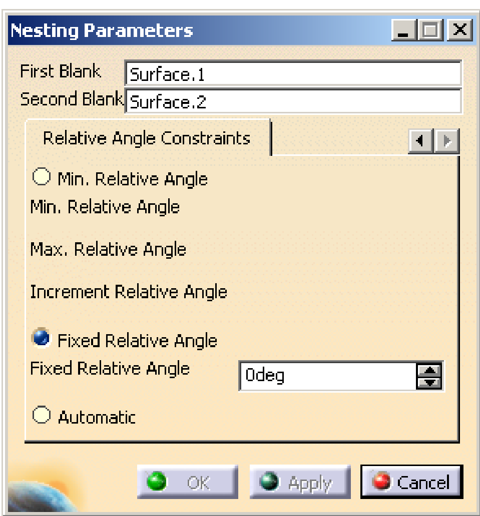
Pitch refers to the relative distance between the blank shapes. If the pitch of the nesting layout is known prior to nesting, this information can be added as a constraint on the **Pitch Constraints** tab of the Nesting Parameters dialog box. If a maximum pitch constraint is added, only the nesting layouts with a pitch equal to or less than the specified value will be calculated.



To control how the first blank is rotated on the coil an angle constraint can be defined on the Layout Angle Constraints tab of the Nesting Parameters dialog box. If a Variable Layout Angle constraint is added, only the nesting layouts that fall within the minimum and maximum values will be calculated. If a Fixed Layout Angle is defined, only nesting layouts with the fixed layout angle value will be calculated. Defining an Automatic constraint will calculate the layout angle for the lowest cost.

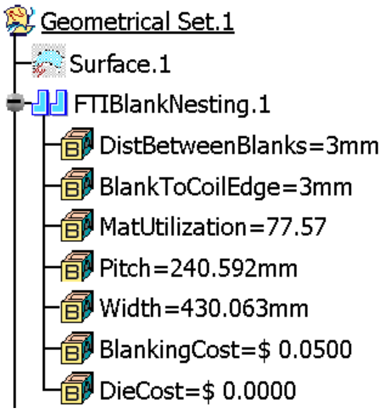


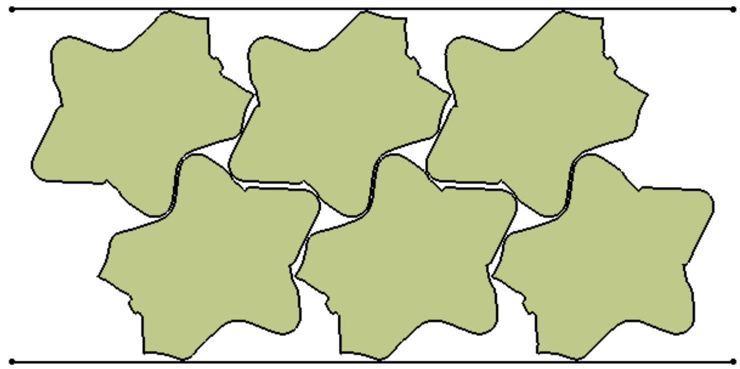
To control how the second blank is rotated on the coil an angle constraint can be defined on the **Relative Layout Angle Constraints** tab of the Nesting Parameters dialog box. If a Variable Relative Angle constraint is added, only the nesting layouts that fall within the minimum and maximum values will be calculated. If a Fixed Relative Angle is defined, only nesting layouts with the fixed relative angle value will be calculated. Defining an Automatic constraint will calculate the relative angle for the lowest cost. Defining a Relative Angle Constraint only is useful for Two Up, Mirror and/or Two Blank layouts.



Viewing Nesting Layout Results

Once the nesting layout calculation is complete the nesting layout with the highest material utilization will be displayed on the X-Y plane and the corresponding result parameters will be listed under the Blank Nesting.1 branch.





MatUtilization: (material utilization) % of the material that is used by the blank

Pitch: calculated relative distance between the blanks

Width: calculated coil width

BlankingCost: represents the cost of each blank based on the Base Material Cost, Extra Material Cost, Scrap Value, Consumables, Gross Weight, and Net Weight.

DieCost: represents the cost of the die per blank

Viewing Other Nesting Layout Results

Additional layout results can be viewed in the Results Table. To open the Results Table, right click on the **FTIBlankNesting.1** branch and select Table Results.

Line	Material Utilization	Blanking Cost	Die Cost	Total Cost	Pitch	Width	Layout Angle	Type Of Layout	Gross Weight
<...	71.28%	0.2747 USD	0.2714 USD	0.5461 USD	166.36mm	176.408mm	89.788deg	One Up	0.343kg
2	71.27%	0.2747 USD	0.2714 USD	0.5461 USD	166.409mm	176.365mm	-0.025deg	One Up	0.343kg
3	71.27%	0.2747 USD	0.2714 USD	0.5462 USD	166.415mm	176.377mm	-0.733deg	One Up	0.343kg
4	71.26%	0.2747 USD	0.2714 USD	0.5462 USD	166.389mm	176.414mm	90.957deg	One Up	0.343kg
5	71.26%	0.2748 USD	0.2714 USD	0.5462 USD	166.42mm	176.398mm	0.478deg	One Up	0.343kg
6	71.25%	0.2748 USD	0.2714 USD	0.5462 USD	166.378mm	176.447mm	89.037deg	One Up	0.343kg
7	71.25%	0.2748 USD	0.2714 USD	0.5462 USD	166.398mm	176.427mm	91.114deg	One Up	0.343kg
8	71.23%	0.2749 USD	0.2714 USD	0.5463 USD	166.403mm	176.477mm	-87.017deg	One Up	0.344kg
9	71.23%	0.2749 USD	0.2714 USD	0.5463 USD	166.479mm	176.407mm	3.209deg	One Up	0.344kg
10	71.22%	0.2749 USD	0.2714 USD	0.5463 USD	166.461mm	176.435mm	87.447deg	One Up	0.344kg

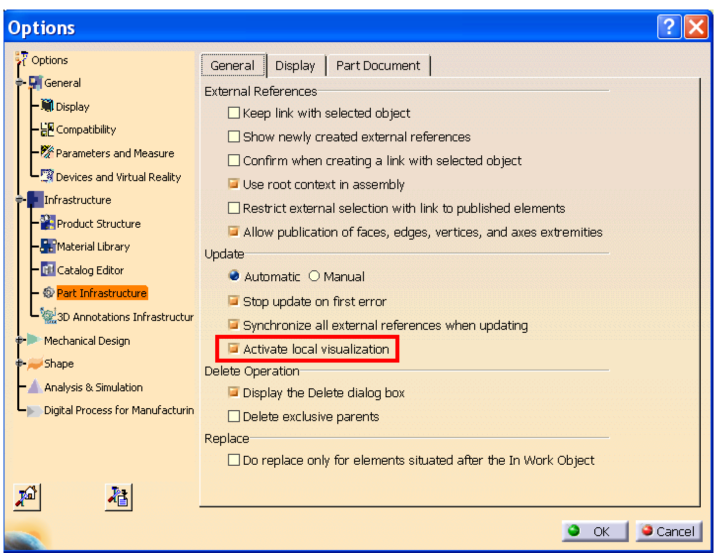
To view other layout results, double click on the row for that result in the Results Table.

<Line#> is the currently displayed layout. Columns can be sorted by selecting the column header.

Nesting Report

When the result is highlighted in the table results, the Report button will become active. Select Report, choose the format of the report, either HTML or Excel based, name the report and select Save.

The report will automatically open once created. When selecting the Excel based report you may need to enable the macros to view all the data, or reduce your security settings in Excel.



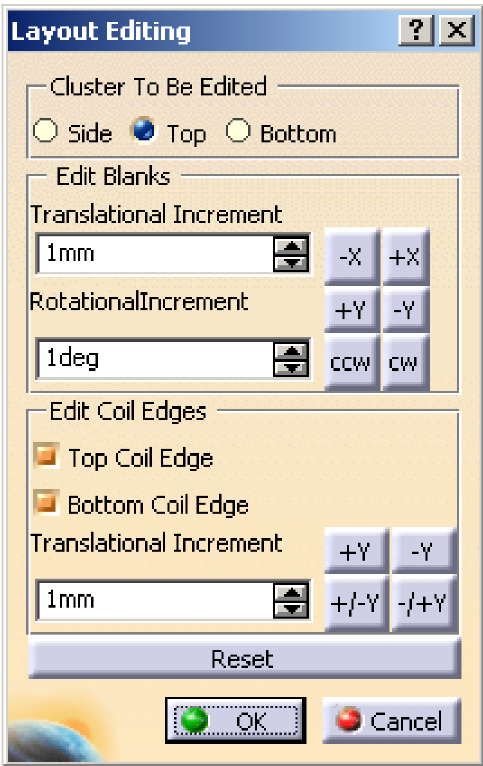
NOTE: In order to show images on the HTML report, you must turn on Activate local visualization on the General tab under Tools-Options-Infrastructure-Part Infrastructure.

Layout Editing

The nesting layout can be manually manipulated to test any changes or adjustments to the automatically generated nest. Using this feature, changes in blank shape and its effects on nesting results can be tested by overlapping the blanks.

To optimize the blank, right click on the **FTIBlankNesting.1** branch and select Layout Editing. The **Layout Editing** dialog box will open. There are three areas in the Layout Editing dialog box: Cluster To Be Edited, Edit Blanks and Edit Coil Edges.

For One Up layouts the Cluster To Be Edited area is not displayed in the Layout Editing dialog box.



Cluster To Be Edited

Side: When side is chosen, the two blanks on the right will be moved as a group.

Top: When top is chosen, the blanks on the top will be moved as a group.

Bottom: When bottom is chosen, the blanks on the bottom will be moved as a group.

Edit Blanks

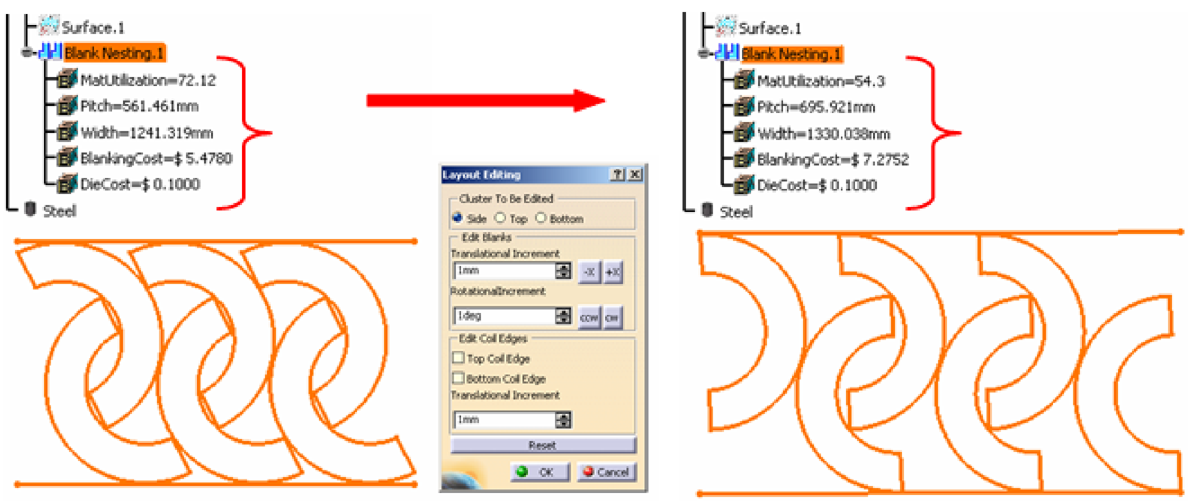
Translation - The blanks can be moved right or left (+/- X direction) by defining an increment of movement and selecting the –X and +X buttons. The blanks can be moved up or down (+/- Y direction) by defining an increment of movement and selecting the –Y and +Y buttons. The +Y/-Y buttons are not visible if the side cluster has been selected.

Rotation - The blanks can be rotated in the counterclockwise (ccw) or clockwise (cw) direction by choosing the increment of rotation (degrees) and selecting the appropriate button.

Edit Coil Edges

The top and/or bottom coil edge can be moved up or down (+/- Y direction) by defining an increment of movement and selecting the –Y and +Y buttons. If both the top and bottom coil edges are selected the +/-Y and -/+Y buttons will be available to move the coil edges at the same time in opposite directions.

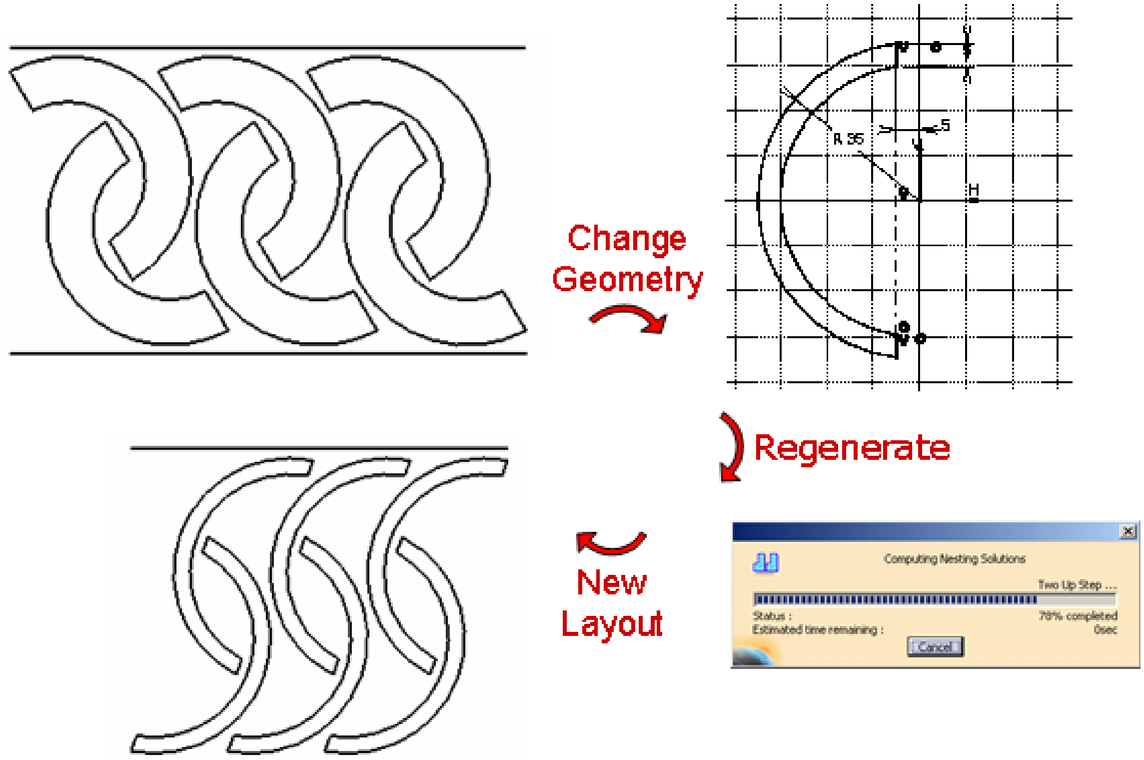
Notes: Clicking on the Reset button will return the blanks to the original nest. The parameters under the Blank Nesting.1 branch associatively update as the nesting layout is modified in the Layout Editing dialog box.



Associative and Regenerative Layout

If you make changes to the nesting layout inputs that affect the results (e.g., geometry, part to edge distance, layout angle, etc.), the results stored under the FTIBlankNesting.1 branch will be recalculated.

This associative link between the solution and all the inputs will prevent any user from viewing data that is no longer valid due to a change implemented between the first time the analysis was run and the point when the user reopens the result for viewing.



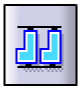
Chapter 7: ProgNest

In this section:

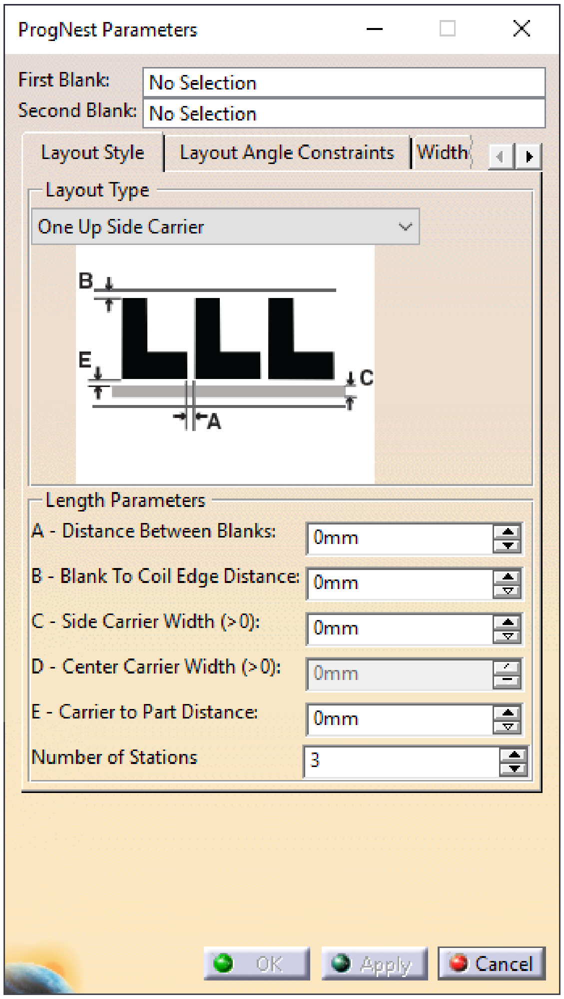
- Progressive Nesting (ProgNest) Parameters
- Layout Style
- Layout Angle Constraints
- Width Constraints
- Progression Constraints
- Relative Angle Constraints
- Layout Editing
- Report Generation

Progressive Nesting Parameters

ProgNest is specifically designed for producing the initial strip layout for progressive die forming operations. For the progressive nesting calculation, some parameters need to be defined. These are entered into the ProgNest Parameters dialog box. To open the ProgNest Parameters dialog box select the Compute ProgNest Layouts icon on the Nesting toolbar.



An **FTIProgNest.1** branch is added to the Specifications Tree and the ProgNest Parameters dialog box opens when the Compute ProgNest Layouts icon is selected.



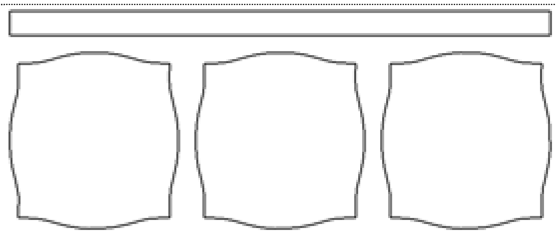
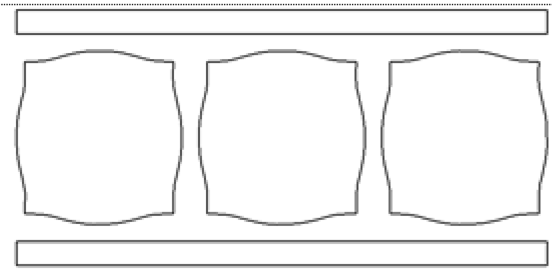
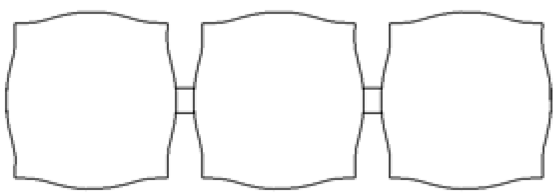
Select the blank surface (Blank Geometry) to define the geometry to be used in the ProgNest layout.

Layout Style

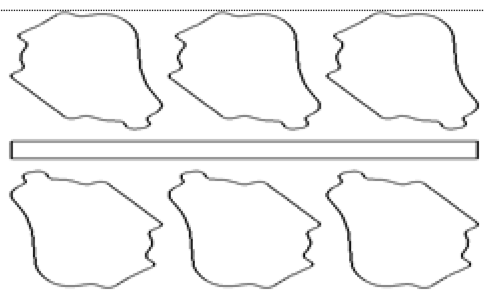
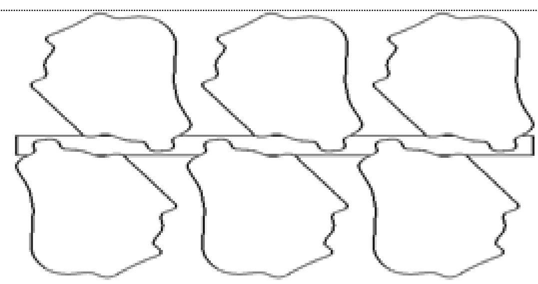
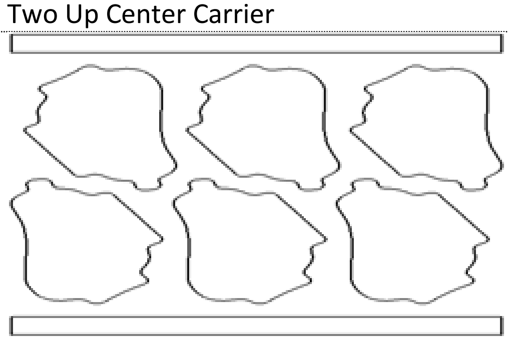
On the Layout Style tab of the ProgNest Parameters dialog box specify the Progressive Nesting layout to be calculated.

There are 3 different layout styles to choose from for Progressive Nesting. Each layout has 3 carrier conditions making a total of 9 possible options.

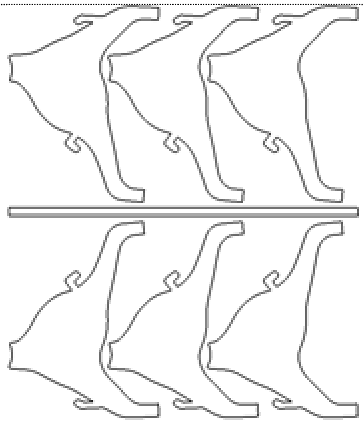
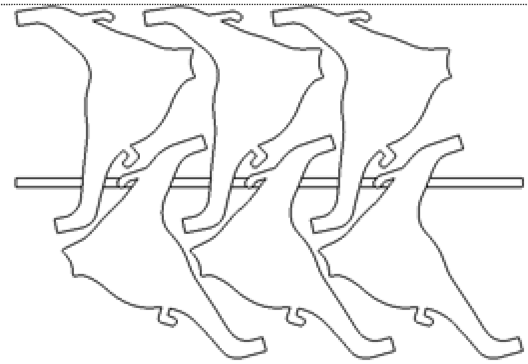
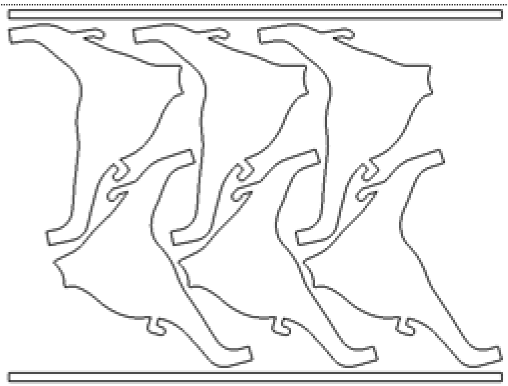
One Up

 One Up Side Carrier	 One Up Double Side Carrier
 One Up Center Carrier	

Two Up

 Two Up Center Carrier	 Two Up Nested Carrier
 Two Up Double Side Carrier	

Mirror

 Mirror Center Carrier	 Mirror Nested Carrier
 Mirror Double Side Carrier	

Layout Parameters

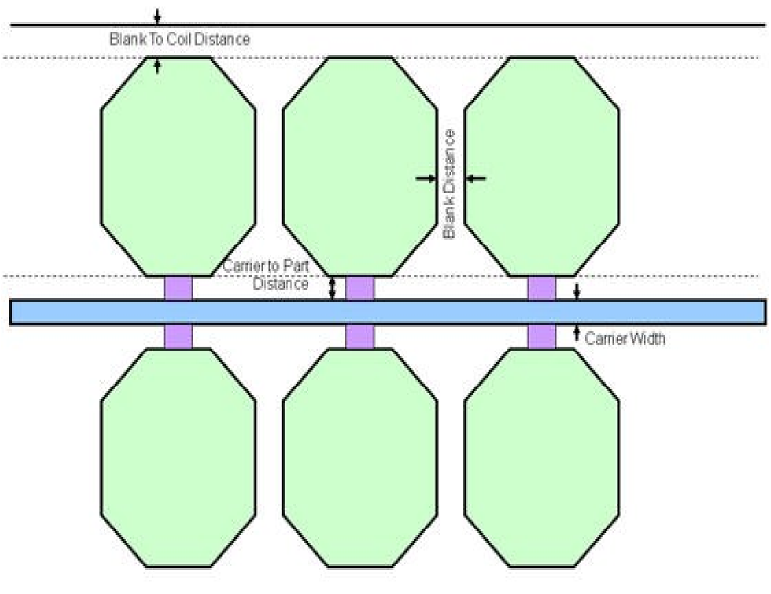
You can also specify the **Distance Between Blanks**, **Blank to Coil Edge Distance**, **Carrier Width and Carrier to Part Distance** parameters.

The **Distance Between Blanks** is the minimum distance between the nested blanks.

The **Blank to Coil Edge Distance** is the minimum distance between the nested blanks and the edge of the coil. This parameter is only used when there is no side carrier at the edge of the coil.

Carrier Width determines the width of the carrier strip that is created in the layout.

Carrier to Part Distance is the minimum distance between carrier and nested blanks. This value is often used to ensure that sufficient space is allowed for stretch webs and other attachment geometry.



By selecting **Create Attachment Surfaces**, the nested layout will be constructed with surfaces connecting the carrier strip to the nested blanks. The location of these connection surfaces cannot be controlled by the user so they should be considered to be for visualization only.

You can also specify the number of stations/blanks that you would like to have for the progressive nest layout. Simply enter the number of stations in the **Number of Stations** field.

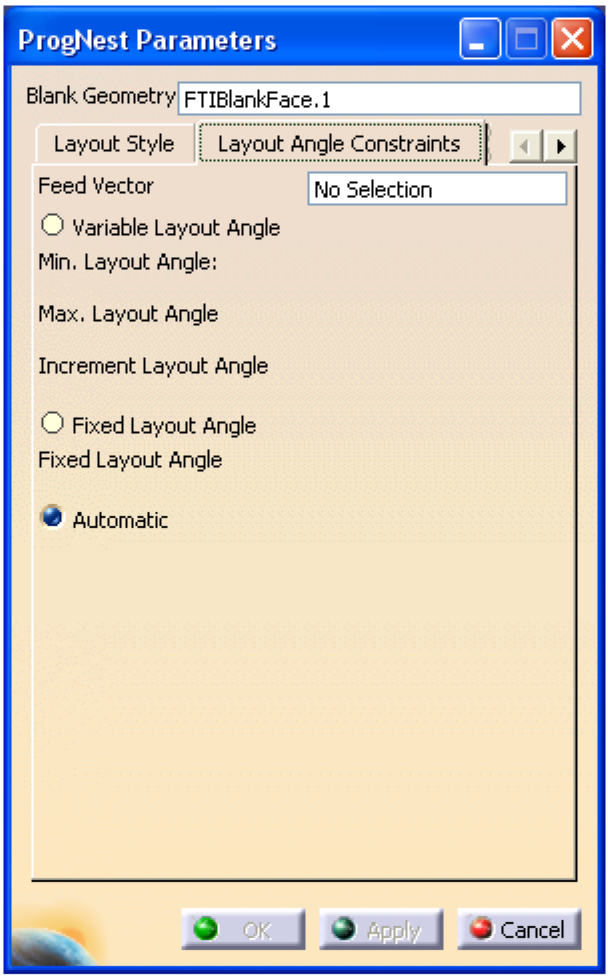
Layout Angle Constraints

The **Layout Angle** determines the orientation of the blank on the coil. ProgNest will initially be set to **Automatic** and generate a table with all blank orientations that fall within the constraints.

For a specific range of rotation angles select **Variable Layout Angle**. Input both the minimum and maximum angle while also selecting the increment angle to adjust the number of cases generated in the table. Results will contain all of the specified increment values as well as additional solutions between the specified increments.

To create a solution with a specific rotation angle, choose **Fixed Layout Angle**. Input the desired angle into the dialog box. If the angle conflicts with any given constraints then the analysis will fail and a ProgNest error will occur.

You can also define a line as a feed vector; this line will be used as the basis for the angles specified in the various constraint parameters. For example, if you want the blanks to be oriented such that a specific line will be lined up with the length of the coil, select the line as the feed vector and set a fixed layout angle of zero degrees.



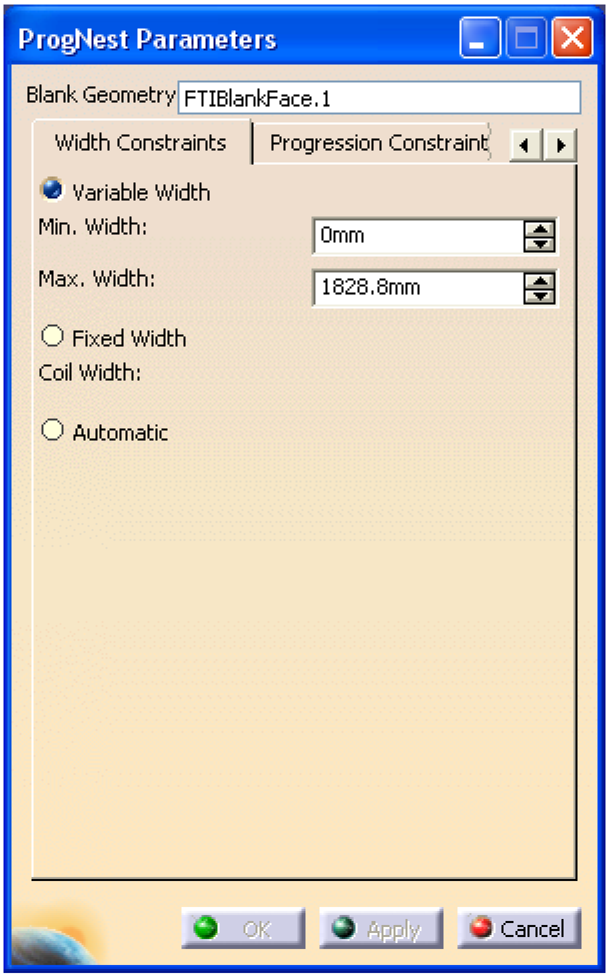
NOTE: The feed vector line must be in plane with the blank.

Width Constraints

The **Width** refers to the width of the coil. ProgNest will initially be set to **Variable Width** with a minimum width of zero (no minimum) and a maximum width of 1828.8mm (72 inches) which is typically the widest coil that can be purchased.

The **Automatic** option will not restrict the solution.

To restrict the solution to a specific coil width, choose **Fixed Width**. Input the desired width into the dialog box. If the width conflicts with any given constraints or is too small to carry the blank then the analysis will fail and a ProgNest error will occur.

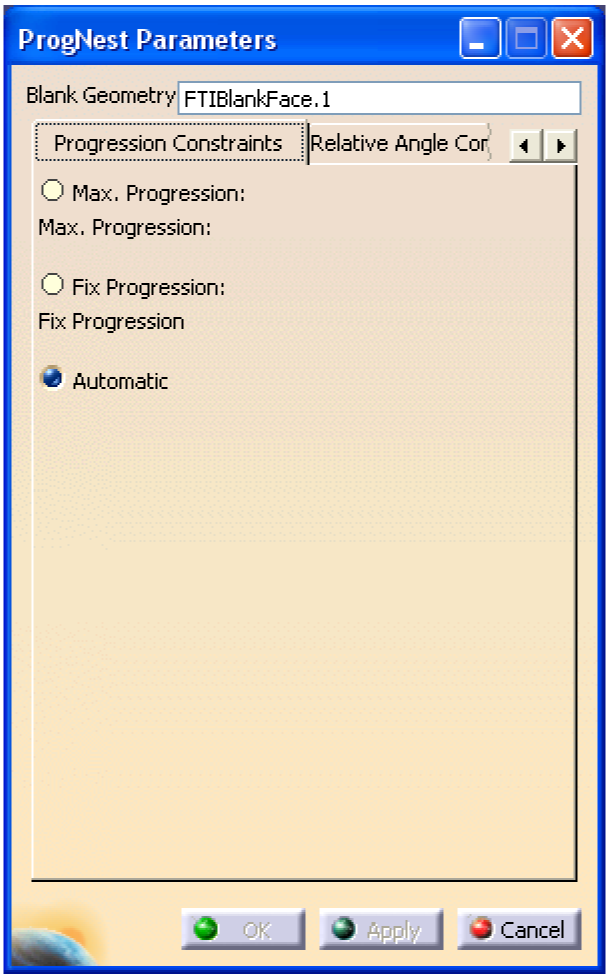


Progression Constraints

Progression is a term used to describe the distance from one part to the next in a progressive die. ProgNest will initially be set to **Automatic** and generate a table with all solutions that fall within the other constraints.

For a specific maximum of progression select **Max. Progression**. Input the largest allowable progression between two blanks.

To restrict the solution to a specific progression, choose **Fix Progression**. Input the desired progression into the dialog box. If the progression conflicts with any given constraints or forces the blanks to overlap each other then the analysis will fail and a ProgNest error will occur.

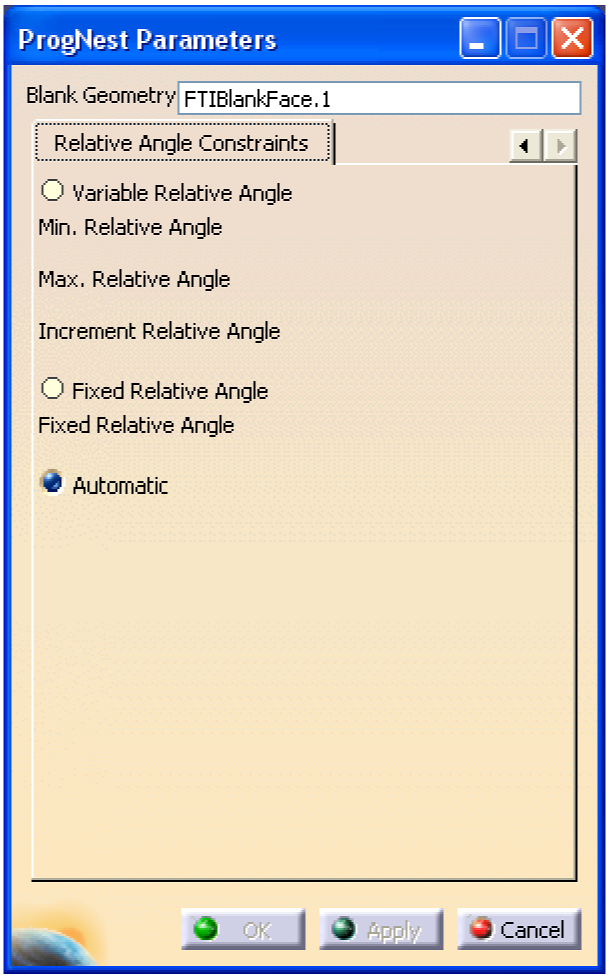


Relative Angle Constraints

The **Relative Angle** only affects mirror layouts. It refers to the angle between the original blank and its mirror. ProgNest will initially be set to **Automatic** and will generate a table with all the solutions that satisfy the other constraints.

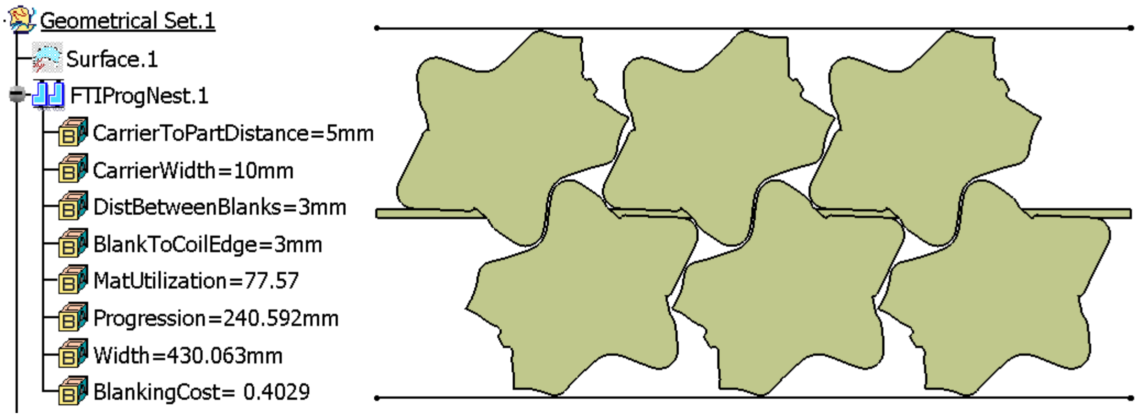
For a specific range of relative angles select **Variable Relative Angle**. Input the minimum and maximum relative angles as well as the desired increment. Results will contain all of the specified increment values as well as additional solutions between the specified increments.

To restrict the solution to a specific relative angle, choose **Fix Relative Angle**. Input the desired angle between the blank and its mirror into the dialog box. If the relative angle conflicts with any given constraints or forces the blanks to overlap on each other then the analysis will fail and a ProgNest error will occur.



Viewing ProgNest Layout Results

Once the ProgNest layout calculation is complete, the layout with the highest material utilization will be displayed on the X-Y plane and the corresponding result parameters will be listed under the FTIProgNest.1 branch.



MatUtilization – (material utilization) % of the material that is used by the blank (as opposed to scrap)

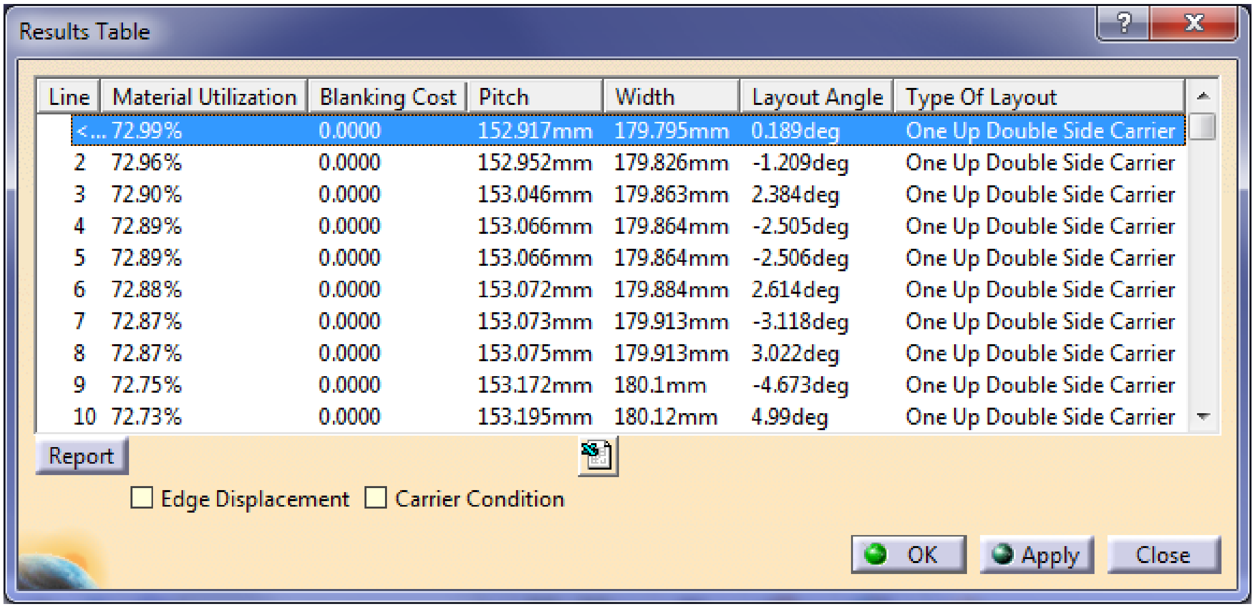
Pitch: calculated distance from one blank to the next blank in the layout (also the distance that the coil would be advanced after each press stroke)

Width: calculated coil width

BlankingCost: represents the cost of each blank based on the Base Material Cost, Extra Material Cost, Scrap Value, Consumables.

Viewing Other ProgNest Layout Results

Additional layout results can be viewed in the Results Table. To open the Results Table, right click on the **FTIProgNest.1** branch and select **Table Results**.



To view other layout results, double click on the row for that result in the Results Table.

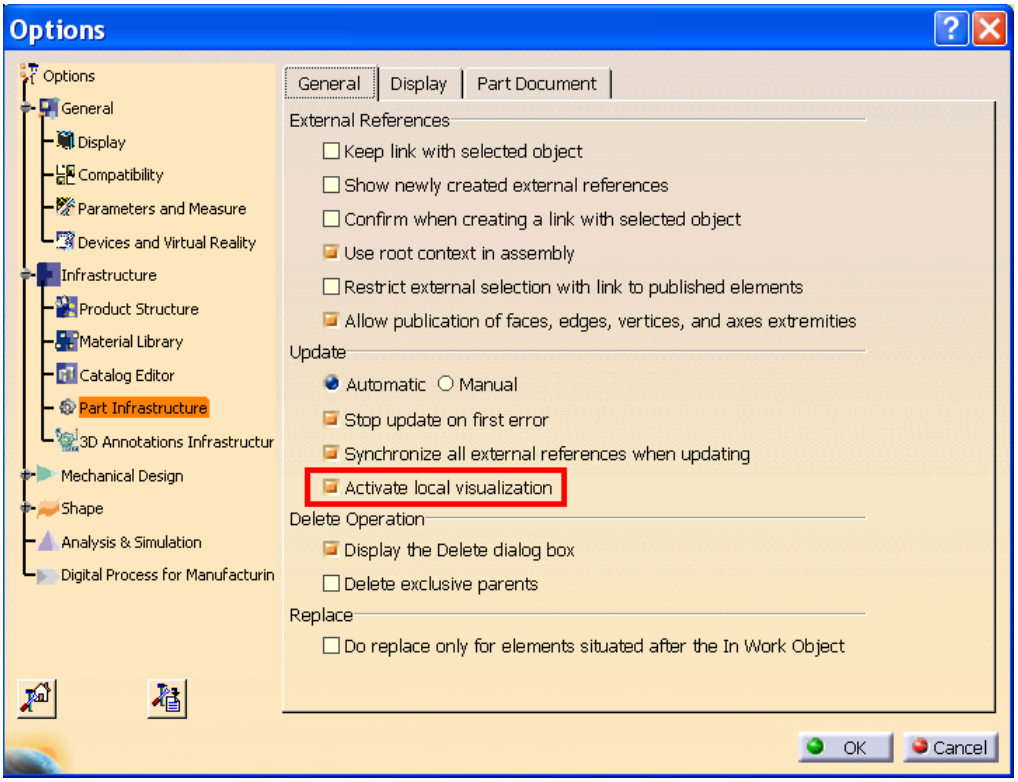
<Line#> is the currently displayed layout. Columns can be sorted by selecting the column header.

Pressing the button with the Excel icon on it will export the contents of the results table as a csv file and open them in Excel.

ProgNest Reports

When the result is highlighted in the table results, the Report button will become active. Select Report, choose the format of the report, either HTML or Excel based, name the report and select Save.

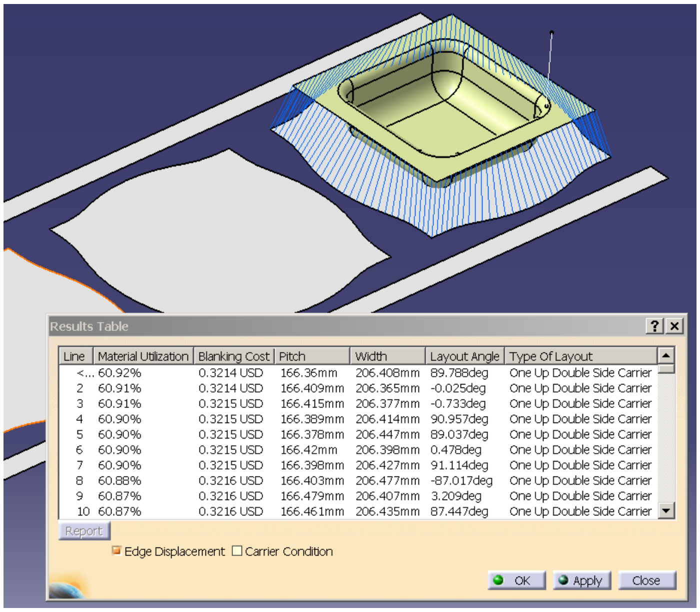
The report will automatically open once created. When selecting the Excel based report you may need to enable the macros to view all the data, or reduce your security settings in Excel.



NOTE: In order to show images on the HTML report, you must turn on **Activate local visualization** on the General tab under Tools-Options-Infrastructure-Part Infrastructure.

Edge Displacement

Check the edge displacement button to display the edge displacement of the part to the blank (shown below).



Carrier Condition

Check the carrier condition button to display a colored contour around the perimeter of the blank displaying whether or not that area is a good edge for a carrier. A green boundary represents an area where a solid carrier or trim flange should be used, a blue boundary represents a stretch web or trim flange, and a red boundary represents a difficult carrier condition.

Editing Parameters

To change the parameters of the progressive nesting layout double click a parameter from the ProgNest feature tree and enter the new value into the dialog box. Only input parameters can be changed from the feature tree. These include Carrier to Part Distance, Carrier Width, Distance between Blanks, and Blank To Coil Edge. The other parameters are results of the nesting calculation and can be referred to when creating formulae in CATIA.

Chapter 8: Cost Optimization

In this section:

- Introduction
- Layout Editing
- Cost Optimization

Introduction

After nesting a blank, you can improve the material utilization of the nest layout by moving the blanks closer together such that they overlap.

Cost Optimizer will then show you where you would have to modify the part geometry in order to realize the material savings.

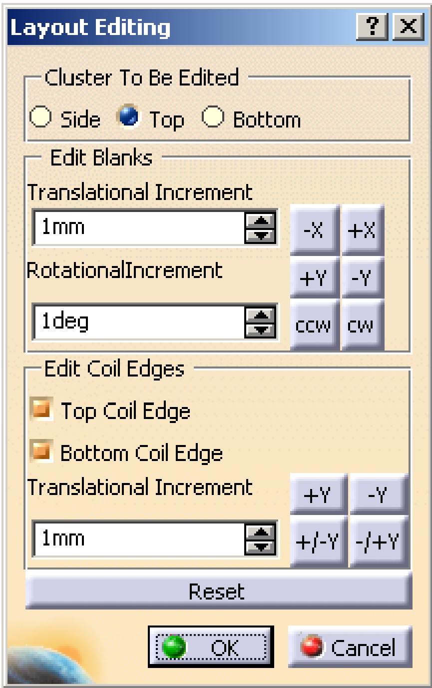
The resulting modified part edge and blank boundary can be used as a guide for modifying the part design in CATIA.

Layout Editing

The nesting layout can be manually manipulated to test any changes or adjustments to the automatically generated nest. Using this feature, changes in blank shape and its effects on nesting results can be tested by overlapping the blanks.

To optimize the blank, right click on the FTIBlankNesting.1 branch and select Layout Editing. The Layout Editing dialog box will open. There are three areas in the Layout Editing dialog box: Cluster To Be Edited, Edit Blanks and Edit Coil Edges.

For One Up layouts the Cluster To Be Edited area is not displayed in the Layout Editing dialog box.



Cluster To Be Edited

Side: When side is chosen, the two blanks on the right will be moved as a group.

Top: When top is chosen, the blanks on the top will be moved as a group.

Bottom: When bottom is chosen, the blanks on the bottom will be moved as a group.

Edit Blanks

Translation: The blanks can be moved right or left (+/- X direction) by defining an increment of movement and selecting the –X and +X buttons. The blanks can be moved up or down (+/- Y direction) by defining an increment of movement and selecting the –Y and +Y buttons. The +Y/-Y buttons are not visible if the side cluster has been selected.

Rotation: The blanks can be rotated in the counterclockwise (ccw) or clockwise (cw) direction by choosing the increment of rotation (degrees) and selecting the appropriate button.

Edit Coil Edges

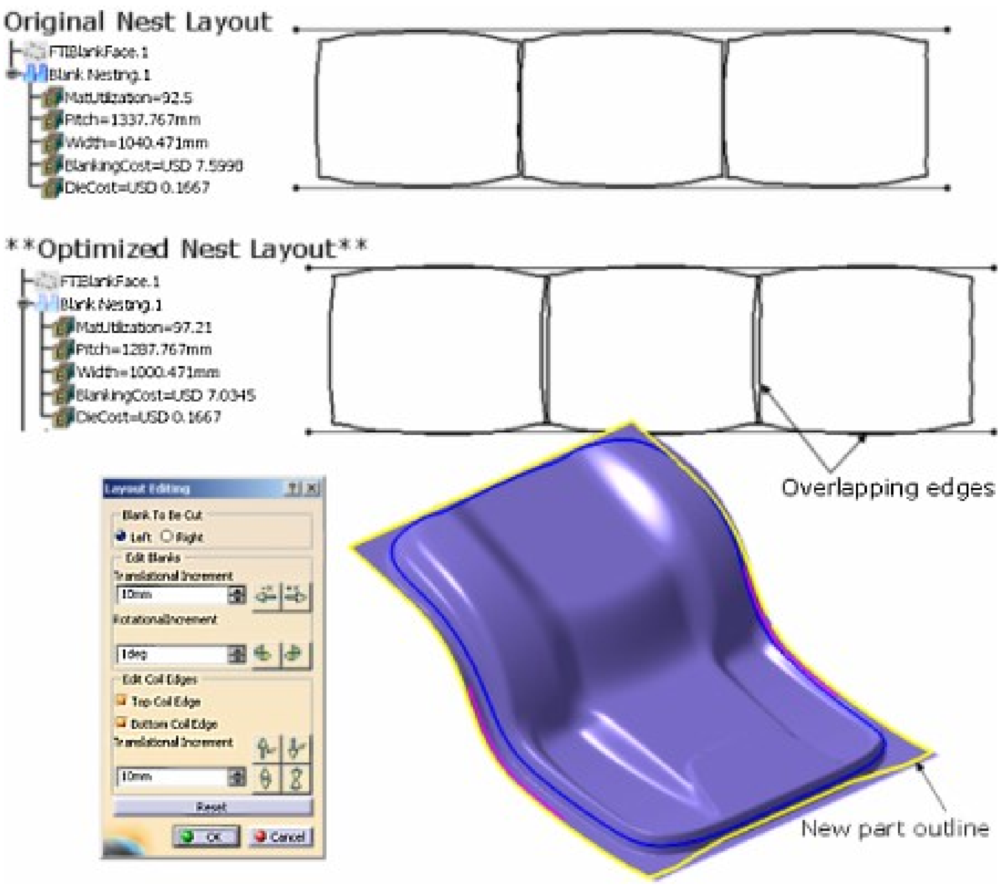
The top and/or bottom coil edge can be moved up or down (+/- Y direction) by defining an increment of movement and selecting the –Y and +Y buttons. If both the top and bottom coil edges are selected the +/-Y and -/+Y buttons will be available to move the coil edges at the same time in opposite directions.

Notes: Clicking on the Reset button will return the blanks to the original nest. The parameters under the FTIBlankNesting.1 branch associatively update as the nesting layout is modified in the Layout Editing dialog box.

Cost Optimization

Using the Layout Editing tools to overlap adjacent blanks or overlap the coil edge and blank edge (see chapter on **Layout Editing**) will result in an updated outline on the part geometry.

The parameters under the **FTIBlankNesting.1** branch will also associatively update as the nesting layout is modified in the Layout Editing dialog box.



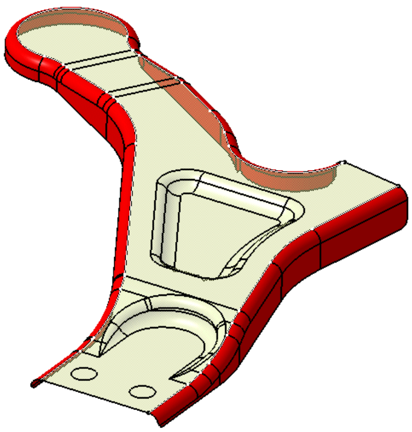
Chapter 9: Trim Line Development

In this section:

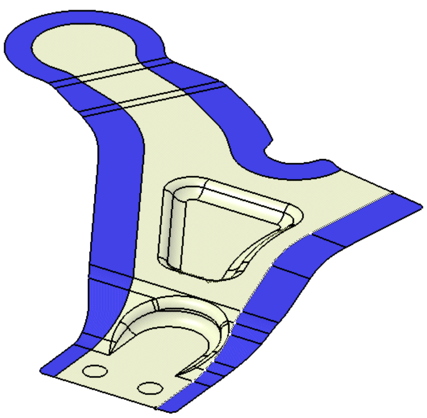
- Introduction
- Model preparation
- Calculating the trim line by means of the function flange unfolding (Trim Line Development)

Introduction

FTI TLD for CATIA V5.3 allows to unfold the flange, in order to, e.g. to derive a blank geometry based on the corresponding drawing geometry. The flange unfolding in FTI TLD is carried out by a finite element analysis and it takes the mechanical features and the displacement of the neutral axis due to the bending procedure into account.



The flanges to be unfolded are shown in red



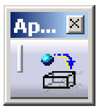
The unfolded flanges are shown in blue

Furthermore, FTI TLD for CATIA contains tools for an automatic determination of a drawing direction, for the calculation of trimming angles and for the determination and evaluation of a cam direction. The respective functions are described in detail in the following sections.

Model preparation

As already mentioned, the mechanical features of the applied material are taken into account concerning the calculation of the unfolding of the flanges / trimming line. Since the material data provided by CATIA do not contain the necessary material features to perform these calculations, FTI refers to its provided material library in **Tools / Options / Infra¬structure / Material Library**.

After selecting the corresponding material library (installation directory / FTICATIAV5RXX / FTICatalog.CATMaterial), a list of materials is provided. To assign a material to a part, the user needs to click on the Apply Material symbol in the lower toolbar.



In the dialog box a material can be selected from the preset list and applied to the part. This assignation occurs by selecting the first entry of the structure tree, therefore , the whole part is selected. This procedure should be preferred because thus, the application of the material is clearly shown evident at the lower end of the structure tree. Alternatively, the flange / s can be selected separately.

Since the sheet metal thickness has a direct impact on the longitudinal expansion of the flange during the unfolding process, it is furthermore necessary to define the applied sheet metal material's thickness. Therefore, different possibilities exist:

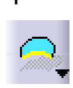
- At the Generative Shape Design (GSD) Workbench in Tools / Thin Parts Attribute Definition or
- At the Part Design Workbench in Thick Surface or Shell Entit.

Within the different functions of FTI TLD a punch direction (Press Direction) needs to be often defined. This may be achieved in various ways:

- Selecting an already existing line
- Selecting a plane surface. In this case, the direction of punch movement is considered as perpendicular direction.
- Selecting a coordinate system. The z-axis is considered as the direction of punch movement.

Furthermore, the flange needs to present a separate geometry. If the flange is part of the part, it first needs to be disconnected from the remaining part geometry. The easiest way is to use the Split function. For that the dividing line between the part and the flange needs to be selected within the Multiple Extract function (in the Dropdown of the Boundary function). Following this, the flange can be divided from the part (tip: By using the option "Keep Both Sides" the part and the flange stay visible and are available for next steps).

Symbols and icons in this step in the Generative Shape Design (GSD):

- Boundary function 
- Multiple Extract function 

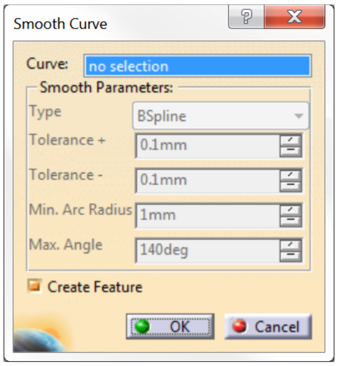
- Split function 

The unfolding of the flange is based on the by FTI developed solver technology and takes the displacement of the neutral axis during the forming process into account. Since FTI TLD is a construction assisting application, it does not provide any possibilities to illustrate metal forming phenomenon like thinning or stretching in the sheet metal’s upper or lower side. Moreover, all steps required for the finite elements analysis (meshing, setting up boundary conditions, etc.) are automated and run in the background.

Smoothing the trim line

(Smooth Curve) 

By means of “Smooth Curve“, a smoothing of the determined curve is possible. By clicking on the icon “Smooth Curve“ in the “Trim Line Development“ toolbar, the curve is smoothened.



The settings are:

- **Curve** enables to select a line / curve which should be smoothened. If it is a 3D-curve, then it can only be smoothened by the B-spline option. If the trim curve is projected in advance by the function “Projection“ from a 3D-polyline to a 2D-polyline , i. e. to the tool layer (XY Plane), then, besides the B-spline option, additionally, there are lines, or lines and circular arcs available.
- **Smooth Parameters**: enable to take settings concerning the smoothing of the trim line. The black points on the trim line indicate the transition from one element to another (e. g. from a line to a circular arc).
  - **Type**: Here, the composition of the trim line is defined. It may consist of 3D-polylines (so-called Basis-splines (Bspline)), lines (lines), or lines and circular arcs (Lines & Arcs).
  - **Tolerance +**:describes the positively tolerated deviation of the smoothened trim line from the original trim line in x- and y-direction.
  - **Tolerance –** :describes the negatively tolerated deviation of the smoothened trim line from the original trim line in x- and y-direction.
  - **Min. Arc Radius**: defines the minimal radius of the used circular arcs.
  - **Max. Angle**: specifies the maximum angle between two elements of the trim line.
- **Create Feature**: the selection of this option enables after closing by OK the generation of an entry of the dialog box ‘Smooth Curve‘ in the tree structure (‘FTIBlankFace‘). Hence, changes concerning the dialog box, or the trim line, respectively, may be done afterwards. Without the selection of this option, by closing by OK only a curve (Curve) of the smoothened trim line is deposited in the tree structure.

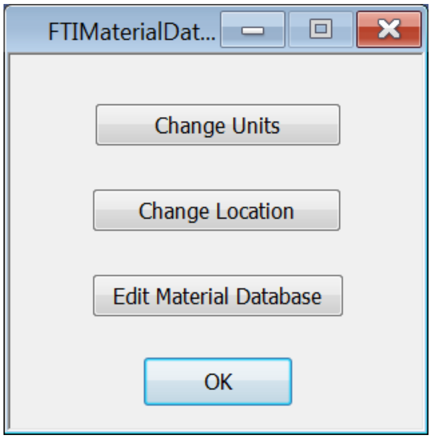
Appendix A – Editing the Materials Database

In this section:

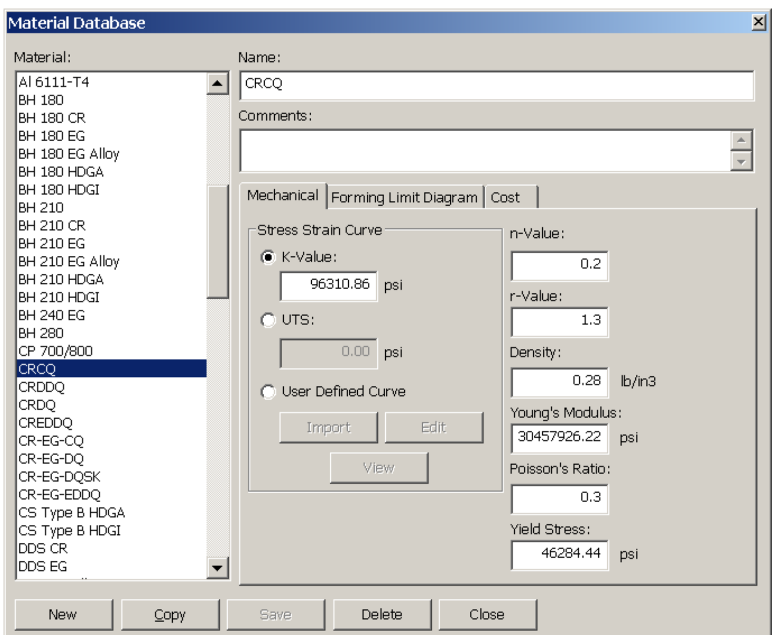
- Editing the FTI Database
- Editing the CATMaterial File

Editing the FTI Database

1. Go to C:\FTI\FTICATIAV5Rxx\dat\matdbeditor
2. Run FTIMaterialDatabase.exe to open the dialog below.



3. Select Edit Material Database to open the Material Database interface shown below.



4. To create a new material select the New button, then enter the required data and then Save the newly created material.
5. If a user defined Stress Strain Curve (SSC) is available, it may be loaded into FTICATIAV5.

   Select a material from the list to add the SSC to, click the Mechanical tab and select the User Defined Curve button, and then select the Import button.

   In the Open dialog window, browse to the folder that contains the stress strain curve file and click Open.

NOTE: The stress-strain curve files are text files (.txt) with the extensions renamed to .ssc. The format of the files is two columns, with strain in the first and stress in the second. The values should be separated by a space. A valid curve will be monotonically increasing when viewed as true stress/true strain. Noise in measured values and drop off at failure should be removed from the data set.

When you open the stress-strain curve data you will be prompted to define the units used in your stress-strain curve. Use the drop down menus to select the strain type, strain units, stress units and stress type. Click OK to load the data.

After defining the units and clicking OK to load the data, a graph illustrating the stress strain curve will automatically open.

After viewing the curve close this image and click the Save button.

You can also click Edit to modify the Stress-Strain curve.

6. If a user defined Forming Limit Curve (FLC) is available, it may be loaded into FTICATIAV5. This will override the FLD0 value.

   Select a material from the list to add a FLC to, click the Forming Limit Diagram tab and select the User Defined Curve button and select the Import button.

   In the Open dialog window, browse to the folder that contains the forming limit curve file and click Open.

   When you open the forming limit curve data you will be prompted to define the units used in your forming limit curve. Use the drop down menus to select the strain type and strain units. Click OK to load the data.

   After defining the units and clicking OK to load the data, a graph illustrating the forming limit curve will automatically open.

   After viewing the curve close this image and click the Save button.

   You can also click Edit to modify the Forming Limit Curve.

Adding the New Material to the FTICatalog.CATMaterial File

Open the FTICatalog.CATMaterial file in CATIAV5 (C:\FTI\FTICATIAV5Rxx). It can be opened in the same manner as a CATPart file. The easiest way to create a new material is to copy an old one and then edit the properties and name:

1. Copy one of the existing materials and paste it into the material set
2. Rename the new material with the required name (it is case sensitive and must match the name of the material in the fti database perfectly)
3. Edit the material properties; right-click on the material and edit the definition of the properties under the Analysis tab. In most cases, the material properties will be taken from the fti database (as opposed to the CATMaterial file), however the density will be used by CATNest and ProgNest for blank weight calculations.