# CATIA V5 FEA

## Release 21

**Jaecheol Koh**
**ONSIA Inc.**

ONSIA

# CATIA V5 FEA Release 21
## A Step by Step Guide

ISBN-13: 978-1470172824
ISBN 10: 1470172824

Author: Jaecheol Koh
Publisher: ONSIA Inc. (www.e-onsia.com)
E-Mail: jckoh@e-onsia.com

# Preface

This textbook explains how to perform Finite Element Analysis using the Generative Structural Analysis workbench in CATIA V5. CATIA is a three dimensional CAD/CAM/CAE software developed by Dassault Systèms, France. This textbook is based on CATIA V5 Release 21. Users of earlier releases can use this book with minor modifications.

It is assumed that readers of this textbook are familiar with creating parts and assemblies in CATIA V5. However, any persons not familiar with CATIA V5 modeling and assembly but interested in FEA can learn through the step by step processes laid out in this textbook, such as naming a part file, creating a 3D model for analysis or defining an FE model. Each process is accompanied by illustrations.

Each chapter deals with a major topic in FEA and proceeds with an analysis procedure using CATIA V5 Structural Analysis. At the end of each chapter the author explains the meaning of the results and recommends additional topics to be considered.

Engineers and mechanical engineering students are highly recommended to read this textbook to increase their knowledge of FEA by using CATIA V5 Generative Structural Analysis.

**Topics covered in this textbook**

- General concepts of FEA
- Singularity in static analysis
- Effects of fillets and stiffeners
- Bearing loads and reflective symmetry
- Rotational loads and cyclic symmetry
- Use of a coordinate system in defining boundary conditions and loads
- Using two dimensional and one dimensional elements
- Connections: Seam weld, rigid, bolt, pressure fit and contact
- Applying loads with enforced displacement
- Automatic mesh adaptation
- Using the temperature effect in static analysis
- Buckling and normal mode analysis

## About the Author

The author is a technical director in ONSIA, Korea and an Affiliate Professor of Songdam College. He has taught in various universities in Korea including Hongik University, Seoul National University of Science and Technologies, Jeonju University and Chosun University. He has more than 15 years of experience in the industrial field of Computer Aided Design and Engineering, teaching engineers courses on three dimensional CAD and CAE software such as CATIA, Siemens NX and SolidWorks. His clients include major companies such as Samsung Electronics, Hyundai Heavy Industries and LG Electronics.

## About ONSIA Inc.

ONSIA is a Korean company specializing in CAD/CAE engineering, education and publication. ONSIA provides engineering services including mechanical design, structural analysis and vibration analysis for companies that manufacture mechanical parts and products. We publish 3D CAD and CAE textbooks in Korean and English.

## Technical Support

For any queries relating to the contents of this textbook, please contact the author at jckoh@e-onsia.com.

# Contents


| | |
|---|---|
| Ch 1: Understanding FEA | 1 |
| Ch 2: Cantilevered Beam | 17 |
| Ch 3: Static Equilibrium and Singularity | 43 |
| Ch 4: Fillet Effect | 63 |
| Ch 5: Stiffener Effect | 81 |
| Ch 6: Bearing Load and Reflective Symmetry | 103 |
| Ch 7: Using a Coordinate System | 127 |
| Ch 8: Centrifugal Force and Cyclic Symmetry | 149 |
| Ch 9: 2D FE Model | 169 |
| Ch 10: Seam Weld and Rigid Connection | 185 |
| Ch 11: Enforced Displacement | 207 |
| Ch 12: Contact and Bolt Connection | 225 |
| Ch 13: 1D FE Model | 251 |
| Ch 14: Pressure Fit and Bolt Connection | 269 |
| Ch 15: Analysis of Alternator Bracket | 299 |
| Ch 16: Automatic Mesh Adaptation | 329 |
| Ch 17: Temperature Effect | 341 |
| Ch 18: Buckling Analysis | 363 |
| Ch 19: Modal Analysis | 379 |

chapter

# 8

# Centrifugal Force and Cyclic Symmetry

## ■ After completing this chapter you will understand

- how to apply rotational velocity and/or acceleration as force.
- the use of cyclic symmetry and how to apply its boundary condition.
- how to use a virtual part to apply a boundary condition.

## 8.1 Introduction

A part that rotates with high speed can break due to centrifugal force. We can take into account angular velocity and/or acceleration in linear static analysis by using **Rotation Force** in the **Loads** toolbar.

If the model for analysis repeats its shape regularly rotating around an axis, it is called a cyclic symmetry model. In this case, we can use only a portion of the model to obtain the same results as using the full model. The theory is similar to the case of reflective symmetry. Note that we have to apply proper boundary conditions on the cut planes.

For the cyclic symmetry model, loads and restraints have to satisfy the cyclic symmetry condition. Centrifugal force and pressure fit are typical examples of cyclic symmetry loads.

The flywheel in Fig 8-1 repeats its shape rotating around an axis. It is under centrifugal load and we will use only 1/8th of the full model. Time and resources for calculation decrease dramatically and we can obtain the same result as when using the full model.



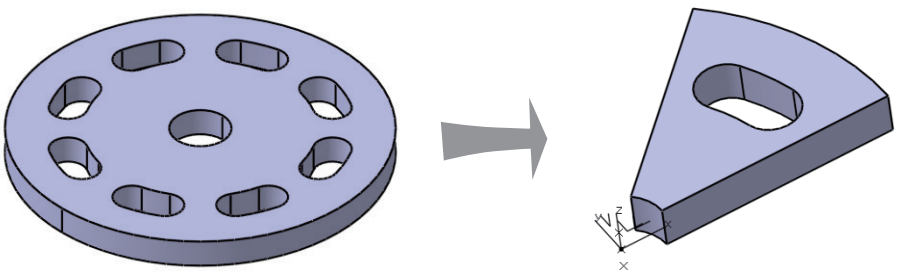


Fig 8-1 Cyclic Symmetry Model

## 8.2 Create 3D Model

***Create a Part***

1. Run CATIA V5 and close all sessions by choosing **File** > **Close** in the menu bar.
2. Choose **File** > **New** and select **Part** from the **New** dialog box.
3. Type in the part name as flywheel_full.CATPart and press **OK**.

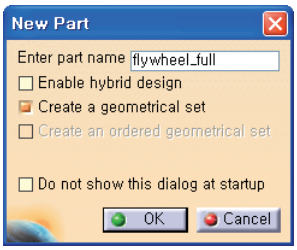



Fig 8-2 Part Naming

## ***Create Full Model***

1. Create a sketch as shown in Fig 8-3 on the xy plane and **Pad** it by 15 mm along both sides. Choose **Mirrored extent** option.
2. Save the part file in d:\catia_cae.

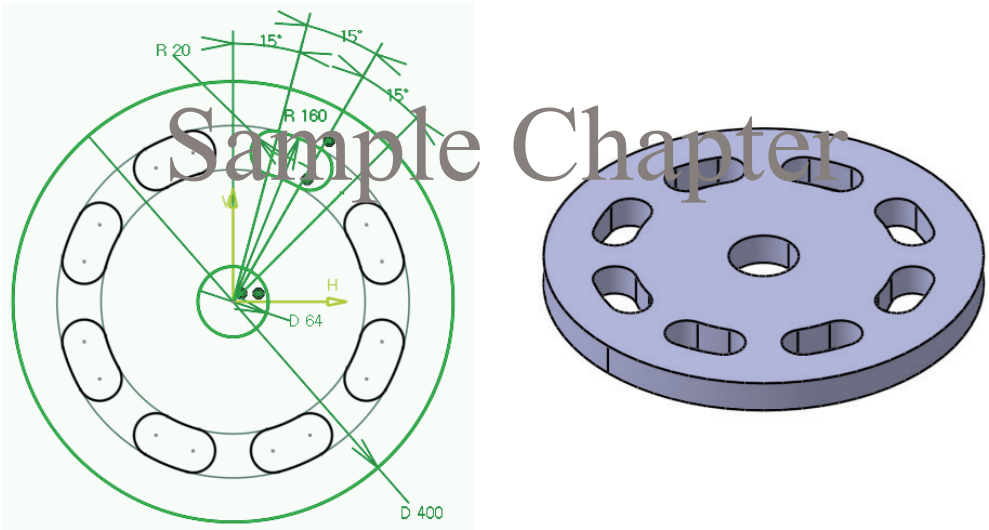





Fig 8-3 Creating a Part (Full Model)

## ***Creating 1/8 Model***

1. Choose **File** > **Save As** from the menu bar.
2. Name the new part as flywheel_1_8.CATPart.

We will analyze the 1/8 model first and then the full model.

## ***Create Reference Plane***

1. Choose the **Axis** button from the **Tools** toolbar.
2. Press **OK** in the dialog box. Coordinate system is created at Global origin (0, 0, 0).

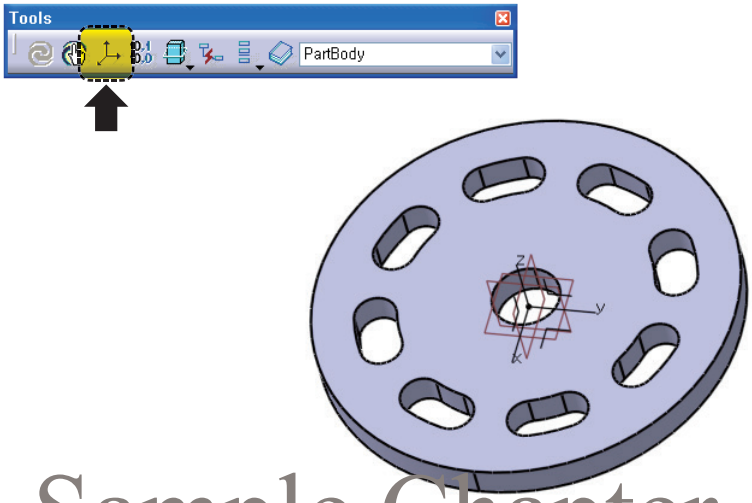



Fig 8-4 Creating Coordinate System



3. Choose the **Plane** button from the **Reference Element** toolbar.
4. Set the **Plane Definition** option. We will create a 45 deg inclined plane with respect to the yz or zx plane.

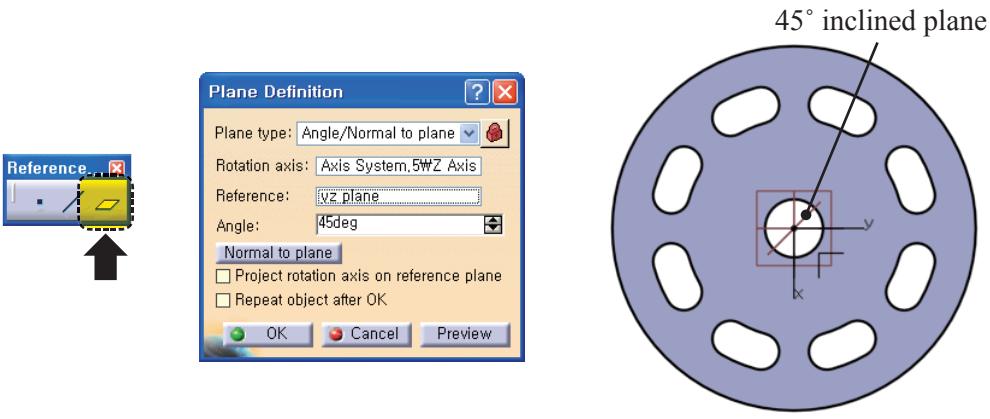



Fig 8-5 Creating Plane

***Split the Model***

1. Choose the **Split** button from the **Surface-Based Features** toolbar.
2. Split the part body with the 45 deg inclined plane as shown in Fig 8-6.
3. Save the file.

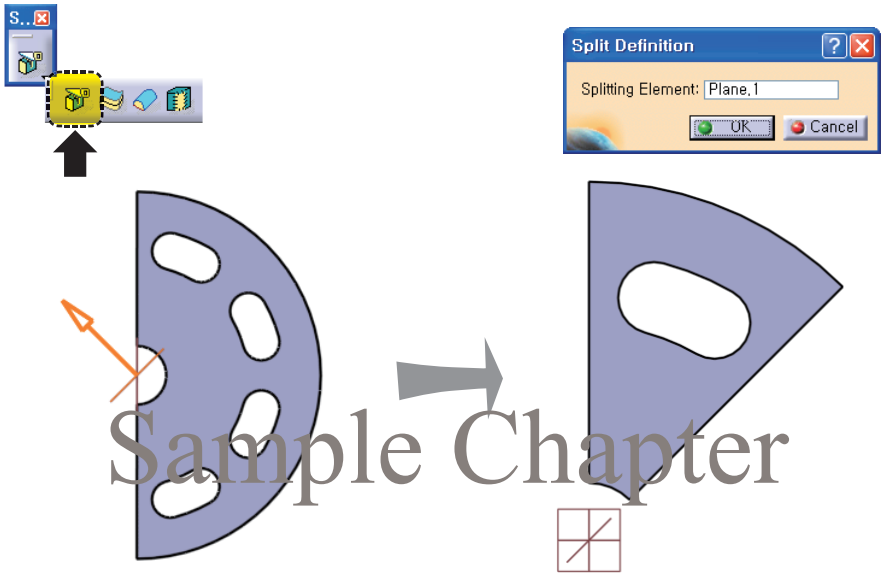





Fig 8-6 Creating 1/8 Model

## 8.3 Mesh Definition

***Execute the GSA Workbench***

1. Press the **Start** button in the menu bar and choose **Analysis & Simulation** > **Generative Structural Analysis**.
2. A warning message appears because we did not apply material for the part. Press **OK** in the **Warnings** dialog box.
3. Select **Static Analysis** in the **New Analysis Case** dialog box and press **OK**.

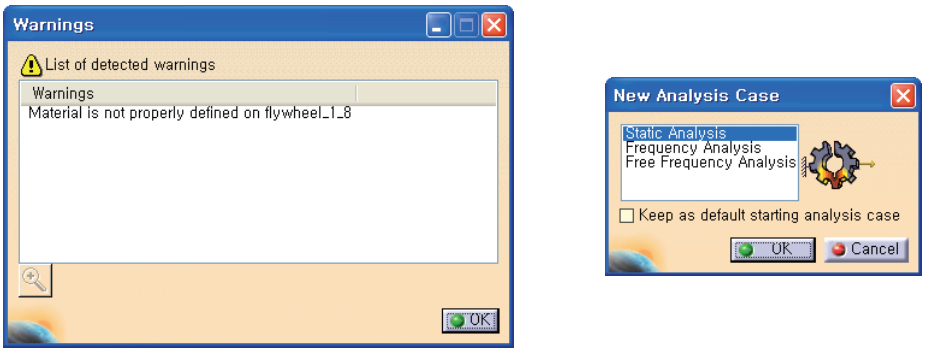



Fig 8-7 Invoking the GSA Workbench

## ***Modify Mesh Definition***

1. Double click on **OCTREE Tetrahedron** of **Nodes and Elements** in the Spec Tree and set the **Size** and **Sag** as shown in Fig 8-8.

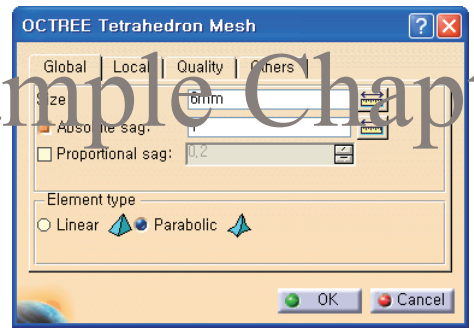





Fig 8-8 Modifying Mesh Option

Let's display the mesh on the geometry. You can skip this step if you are sure about the quality of the mesh.

2. Press mouse button 3(MB3) on **Nodes and Elements** in the Spec Tree and choose **Mesh Visualization**.
3. Read the warning message and press **OK**.
4. Hide the mesh symbol in the model
5. Change the view mode to **Shading with Edges**.

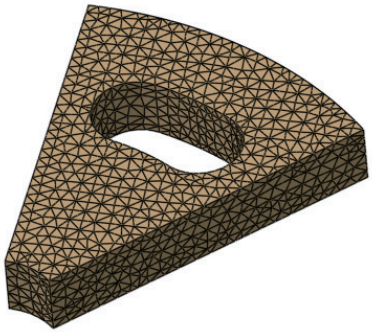


Fig 8-9 Mesh Visualization

## 8.4 Restraints and Loads

Boundary conditions and loads can be applied on 3D geometry such as faces, edges, points and vertices, and the mesh has to be defined on the geometry.

Let's define the symmetric boundary condition on the cut plane.



1. Move the mouse pointer on **Limits Manager.1** and press mouse button 3(MB3). Then choose **Hide/Show** from the pop-up menu to show the 3D geometry.
2. Choose the **Surface Slider** button from the **Restraints** toolbar.
3. Select the plane cut by the zx plane and press **OK** in the dialog box.

**Surface Slider** restraints translation normal to the plane and frees other degrees of freedom. Note that there are no rotational degrees of freedom in the 3D element.

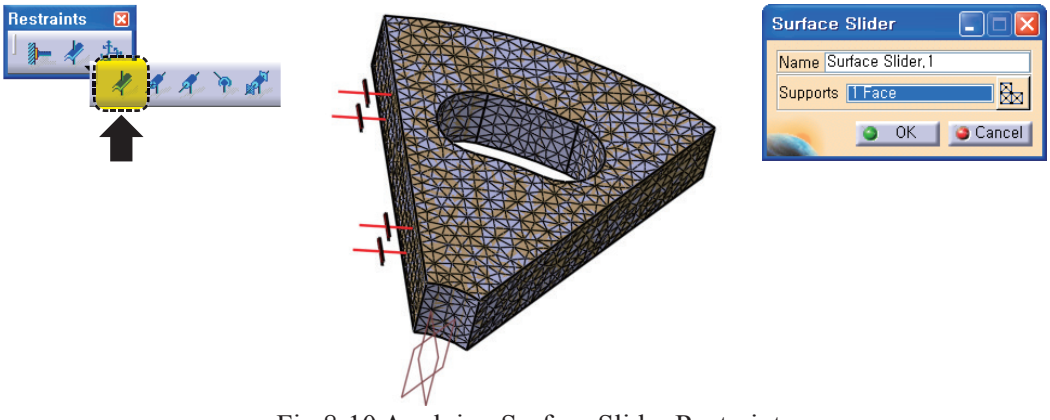



Fig 8-10 Applying Surface Slider Restraint

You can apply **Surface Slider** restraint on the 45 deg slanted plane, but in this example we will use the **User-defined Restraint** function to achieve the same effect. If we are going to restrain the normal direction translation on the slanted plane with **User-defined Restraint**, we have to define the coordinate system first.

Let's define the coordinate system aligned to the inclined plane.

1. Expand Links **Manager.1** in the Spec Tree.
2. Double click on flywheel_1_8 to invoke the **Part Design** workbench.

We cannot use the coordinate system that has been created to define the 45 deg inclined plane because it is not in accordance with the normal direction of the plane.

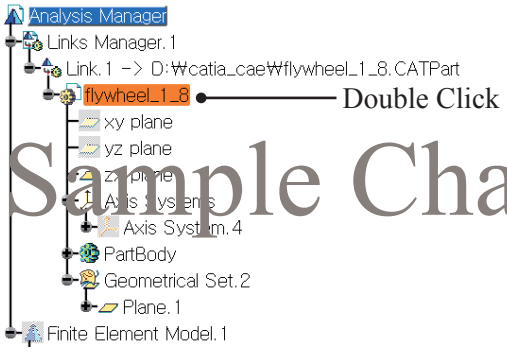





Fig 8-11 Invoking the Part Design Workbench

3. Choose the **Axis** button from the **Tools** toolbar.
4. Show the existing coordinate system in the Spec Tree.
5. Select the origin of the coordinate system.
6. Select the edge on the plane to restrain as the X axis (❻ in Fig 8-12). If the direction is not correct check the **Reverse** option in the dialog box.
7. Select the plane to restrain as the Z axis (❼ in Fig 8-12).

The Y axis is defined automatically. Suppose that the coordinate system is **Left-handed** as shown in Fig 8-12. This may not be your case.

8. Press the **OK** button in the dialog box and hide the first coordinate system.

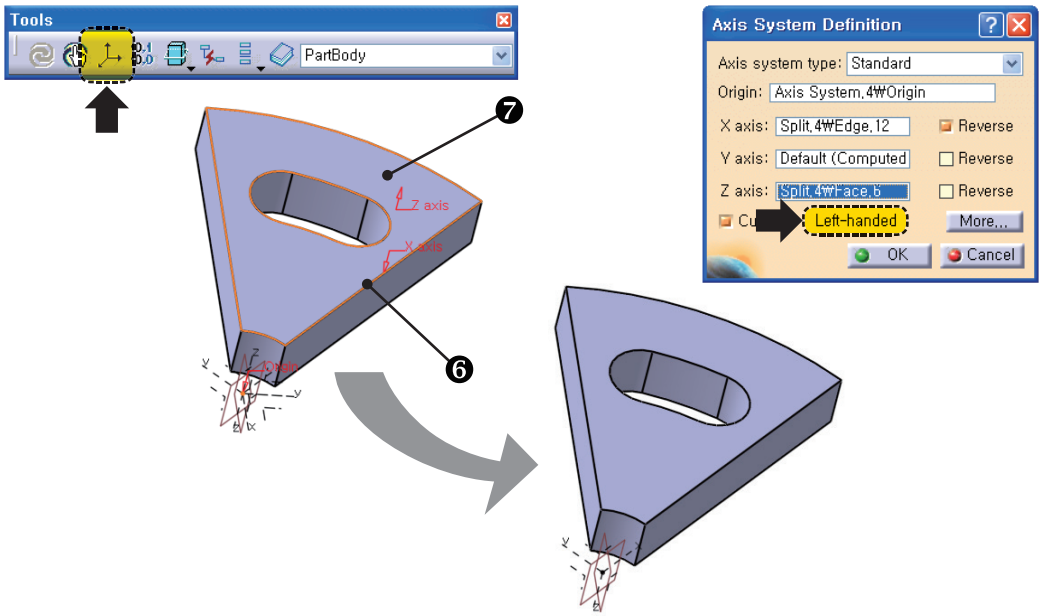



Fig 8-12 Creating Coordinate System

Now, let's define the **User-defined Restraint**.



1. Double click on **Finite Element Model.1** in the Spec Tree to invoke the **GSA** workbench.
2. Show **Links Manager.1**.
3. Choose **User-defined Restraint** from the **Restraints** toolbar.
4. Select the plane **A** which is 45 deg inclined.

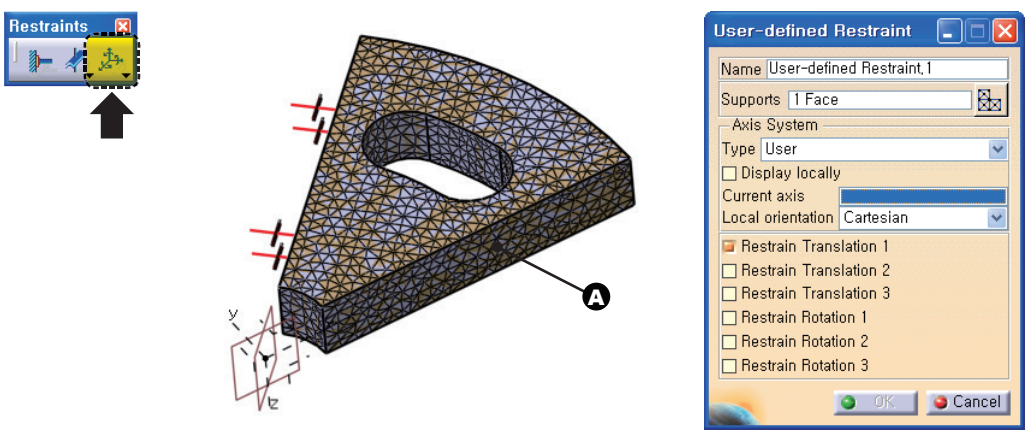



Fig 8-13 Applying Symmetry Restraint

5. Change the **Type** of **Axis System** to **User** in the dialog box and pick **Current axis** selection area.
6. Select the coordinate system that is created in Fig 8-12.

The error message shown in Fig 8-14 appears because the reference coordinate system is not right-handed. Therefore, we have to revert any of the three axes. Close the message.

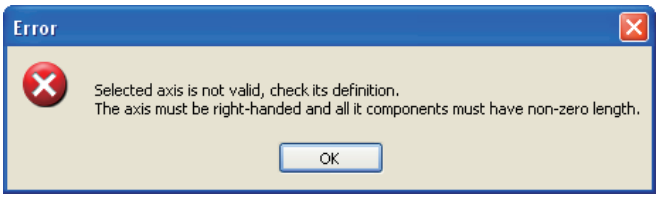



Fig 8-14 Error Message

Let's revert the Z axis of the coordinate system.

1. Expand **Links Manager.1** and double click on **Axis.5** so the **Part Design** workbench is invoked. The suffix number may be different.
2. Double click on **Axis.5** once again in the Spec Tree. The **Axis Definition** dialog box will appear.
3. Check the **Reverse** option next to the **Z axis** selection area.
4. Make sure that the coordinate system is **Right-handed** as in Fig 8-15 and press **OK**.



Now, we can proceed with defining **User-defined Restraint** as shown in Fig 8-16.

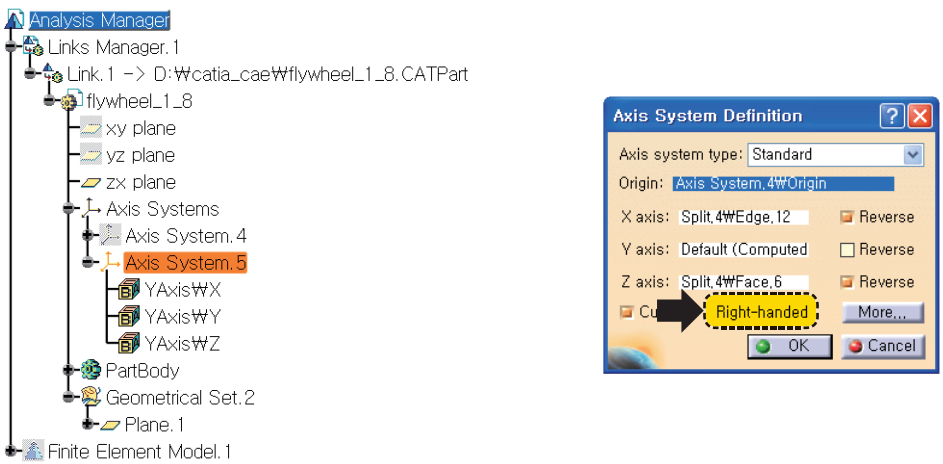



Fig 8-15 Modifying the Z axis

The X, Y and Z axes of the Axis System are recognized as R, θ and Z in the cylindrical coordinate system and each can be restrained using Translation 1, 2 and 3. The rotational degree of freedom of the cylindrical coordinate system can be restrained with Rotation 1, 2 and 3 in the dialog box.

When we define mesh on a three dimensional body, we use the 3D tetrahedron element. Note that the 3D element does not have rotational degree of freedom. Therefore, we do not need to restrain Rotation 1, 2 and 3 in the dialog box. Whether you restrain the rotational degree of freedom of the 3D element or not, the result will not be affected.

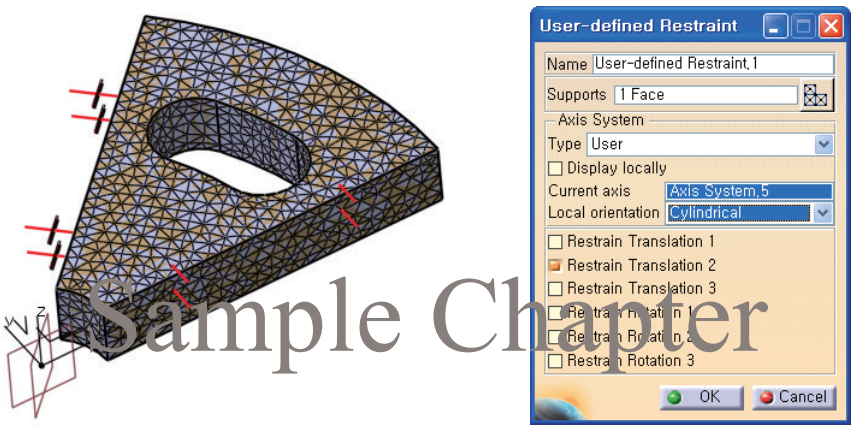





Fig 8-16 Applying User-defined Restraint

## ***Apply Load***

1. Choose the **Rotation Force** button from the **Loads** toolbar.
2. Select **PartBody** as the **Support**.
3. Select Z axis as the **Rotation Axis**.
4. Type in 5000 turn_mn as **Angular Velocity** and press **OK**.

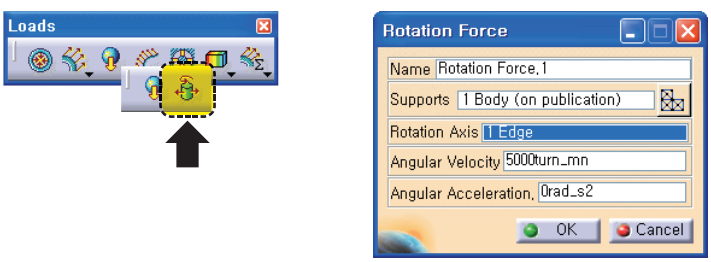



Fig 8-17 Applying Rotation Force

## ***Resolve Singularity***

We did not restrain the Z translational degree of freedom. Therefore, rigid body motion may take place. Let's restrain an arbitrary point in the Z direction.

1. Choose the **User-defined Restraint** button from the **Restraints** toolbar.
2. Select the vertex Ⓐ as shown in Fig 8-18.
3. Restrain the Z Translation with regard to the Global coordinate system as shown in Fig 8-18.

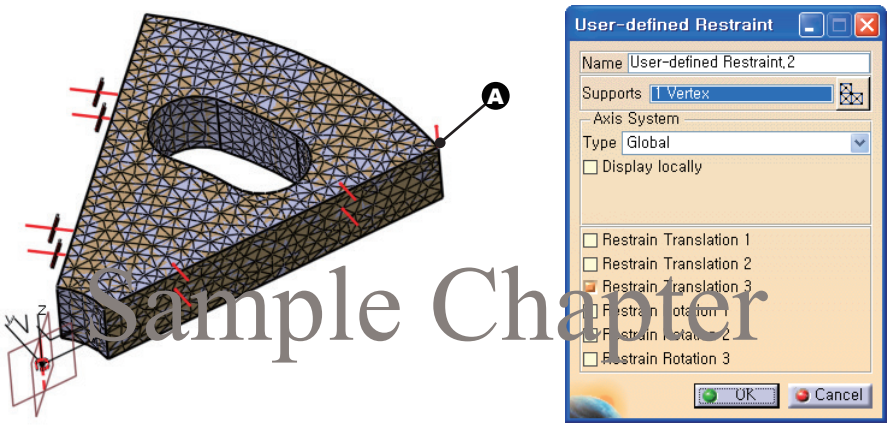





Fig 8-18 Restraining Z Translation

## ***Check the FE Model***

We have to check the FE model to see if the basic requirements were defined successfully.

1. Choose the **Model Checker** button from the **Model Manager** toolbar.
2. Look at the **Status** column from the **Model Checker** dialog box. We have not defined the material yet.

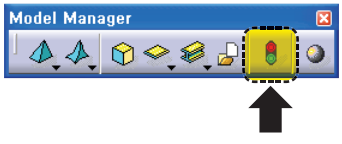



Fig 8-19 Model Check

## ***Define Material***

Let's define the material for the part.

1. Expand **Links Manager.1** and double click on the name of Product (flywheel_1_8). The **Part Design** workbench will be invoked.
2. Press the **Apply Material** button. The **Material Library** dialog box will appear.
3. Select the **Metal** tab and select **Aluminum**. The **OK** button is not activated yet.
4. Select **PartBody** in the Spec Tree and press **OK** in the dialog box.
5. Double click on **Finite Element Model.1** in the Spec Tree. The **GSA** workbench will be invoked.
6. Save the file in d:\catia_cae. The name of the analysis file would be flywheel_1_8. CATAnalysis. Note that the part file is saved at the same time.

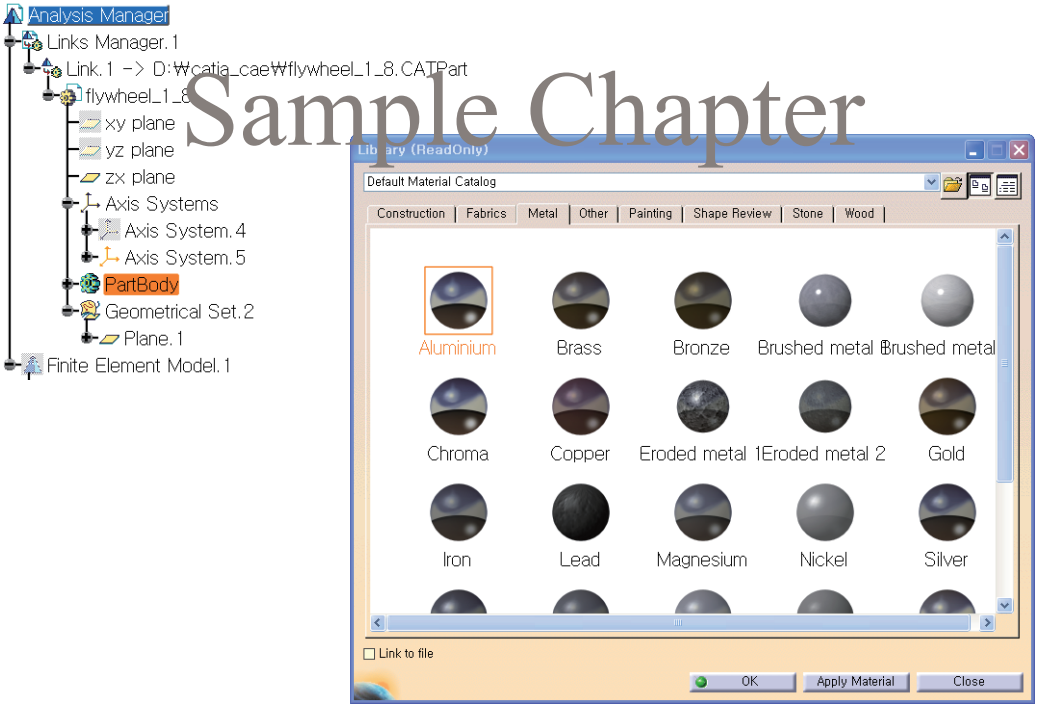





Fig 8-20 Material Library

## 8.5 Solving

Run solver according to the following procedure.

1. Choose the **Compute** button from the toolbar.
2. Set the option to **All** and press **OK** in the dialog box.

The warnings dialog box appears because we applied **Surface Sliding** restraint on the symmetric plane. The **Surface Sliding** restraint restrains rotational degrees of freedom to make sure that normal displacement does not take place. However, there are no rotational degrees of freedom in the 3D element. There is only one message in the dialog box because we applied **User-defined Restraint** for the other symmetric plane.

3. Close the warning message.
4. Resources required for the calculation are displayed in the dialog box. If all the estimations are zero or if the software cannot calculate the estimation, it means that you cannot obtain the results. Press **Yes** and proceed with the solution process.



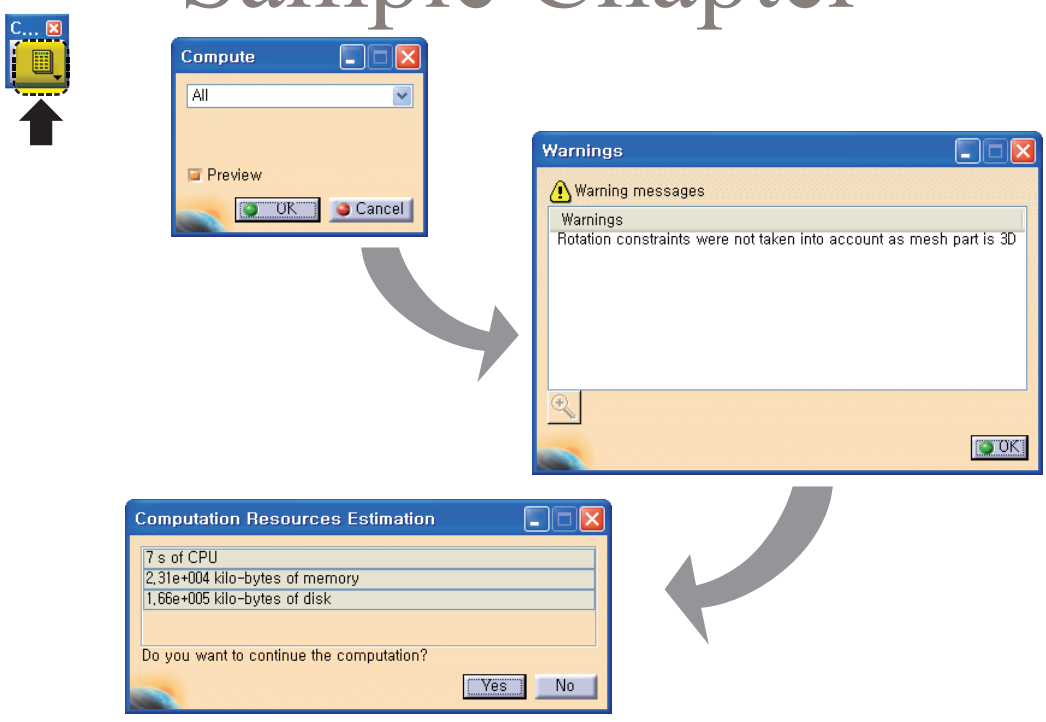



Fig 8-21 Executing Solver

## 8.6 Post Processing

Fig 8-22 shows Von Mises Stress (Average iso) of the 1/8 model. View mode is **Shading with Material**. Save the analysis file.

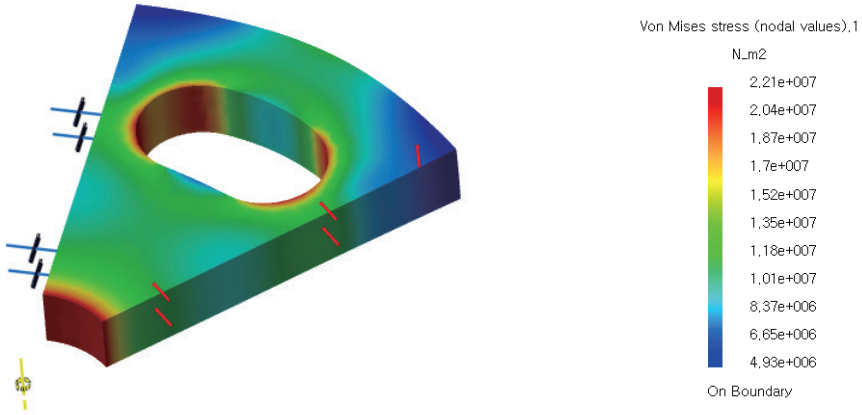



Fig 8-22 Von Mises Stress

## 8.7 Analyzing the Full Model



Let's perform the same analysis with the full model and compare the results.

***Create FE Model***

1. Open flywheel_full.CATPart.
2. Apply the material **Aluminum** to the **PartBody**.

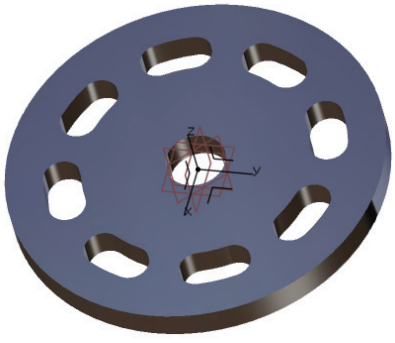


Fig 8-23 Full Model

3. Execute the **GSA** workbench.
4. Select **Static Analysis** in the **New Analysis Case** dialog box and press **OK**.

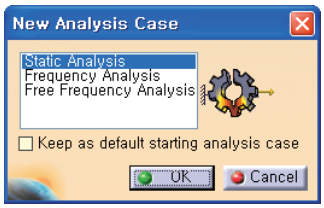



Fig 8-24 Analysis Case

5. Modify the mesh option as shown in Fig 8-25 and visualize the mesh. The mesh size is the same as that of the 1/8 model. Therefore a much larger number of elements is generated. We can expect that the calculation time will take much longer.




Fig 8-25 Modifying Mesh Option

***Restraint and Load***

Let's apply load.

Boundary conditions and loads can be applied on 3D geometry such as faces, edges, points and vertices, and the mesh has to be defined on the geometry.

1. Show **Links Manager.1** in the Spec Tree.
2. Choose the **Rotation Force** button from the **Loads** toolbar.
3. Select **PartBody** as the **Support**.
4. Select the Z axis as the **Rotation Axis**.
5. Type in 5000 turn_mn as **Angular Velocity** and press **OK**.

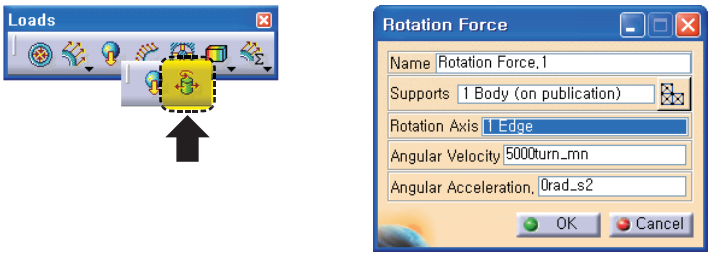



Fig 8-26 Applying Rotation Force

Now, let's think about the restraint.

We have to restrain R, θ and Z translation with regard to the cylindrical coordinate system. Whether you restrain the rotational degree of freedom of a 3D element or not, the result is not affected because there are 3D elements defined.

Let's create a virtual part on the cylindrical surface.

1. Hide **Mesh.1** in the Spec Tree.
2. Expand **Links Manager.1** and hide all planes for convenience.
3. Choose the **Smooth Virtual Part** button from the **Virtual Parts** toolbar.
4. Select the cylindrical surface Ⓐ as the **Supports** and press **OK** in the dialog box. If you do not select an object for **Handler**, it is defined at the center of the support.



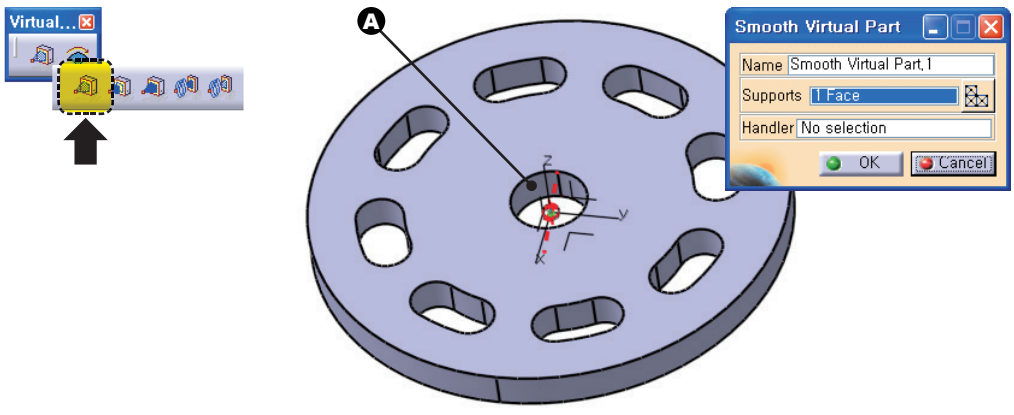



Fig 8-27 Creating a Virtual Part

Now, let's create **User-defined Restraint**.

1. Choose the **User-defined Restraint** button from the **Restraints** toolbar.
2. Select **Virtual Part** as the **Supports**. Be careful not to select the cylindrical face.
3. Set the **Axis Type** to **User**.
4. Pick the selection area of **Current axis** and select the axis system at the center. The suffix number may be different.
5. Choose **Cylindrical** from the **Local orientation** dropdown list.
6. Check each option box to restrain translational degrees of freedom and press **OK**.
7. Save the file in d:\catia_cae. The name of the analysis file would be flywheel_full. CATAnalysis. Note that the part file is saved at the same time.

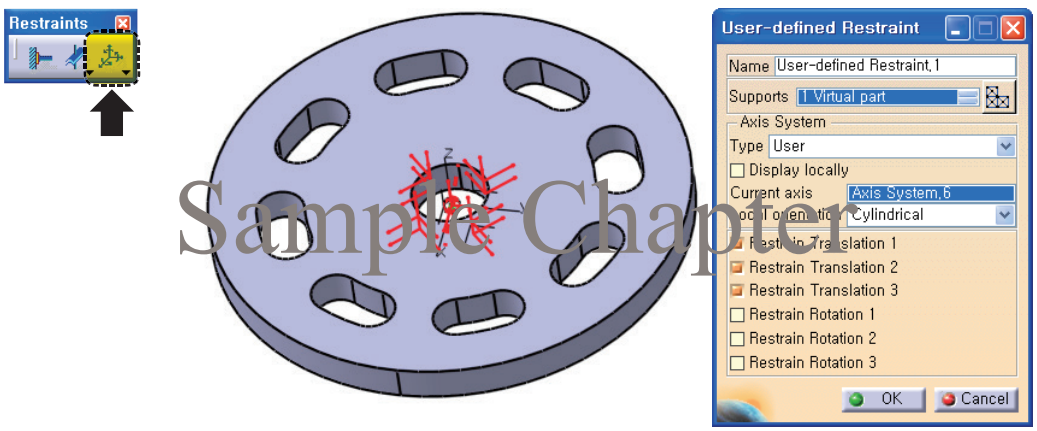





Fig 8-28 Applying User-defined Restraint

***Solving***

1. Choose the **Compute** button from the toolbar.
2. Set the option to **All** and press **OK** in the dialog box.
3. Resources required for the calculation are displayed in the dialog box. If all the estimations are zero or if the software cannot calculate the estimation, it means that you cannot obtain the results. Press **Yes** and proceed with the solution process.

Calculation time takes longer because the number of elements is much higher than when using the 1/8 model.

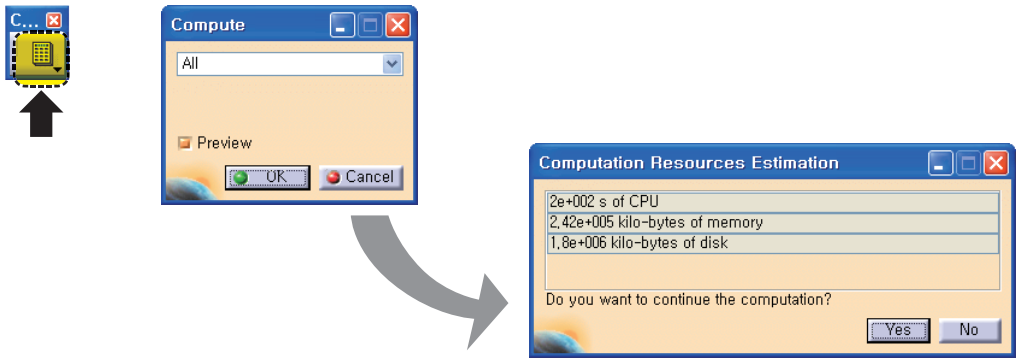



Fig 8-29 Executing Solver

## ***Post Processing***

Fig 8-30 shows Von Mises Stress (Average iso) of the full model. View mode is **Shading with Material**. Save the analysis file.



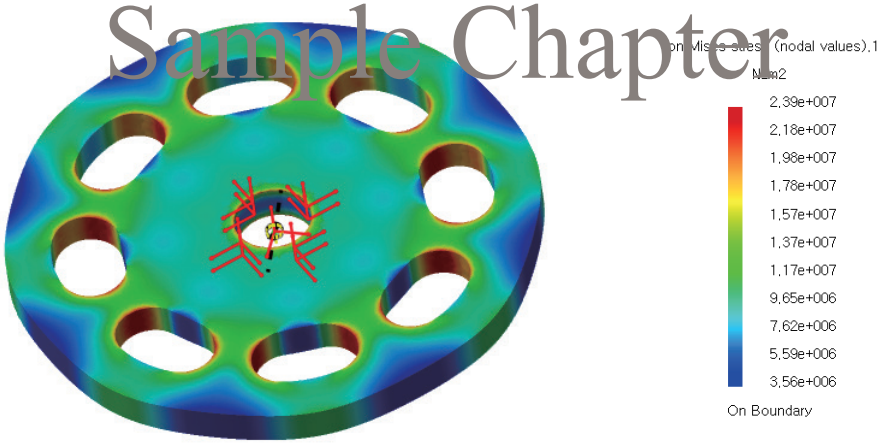
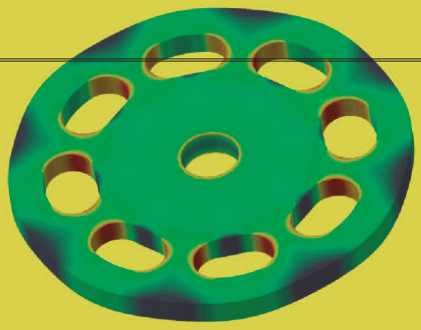


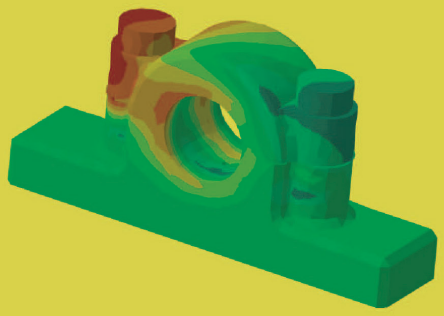

Fig 8-30 Von Mises Stress

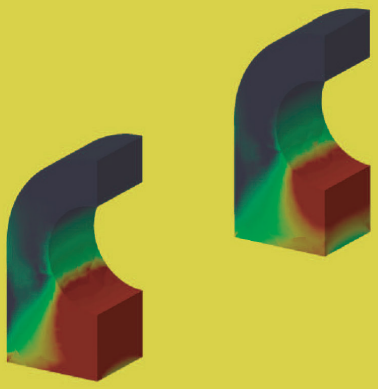
## 8.8 Evaluation of the Results

The results of the full model and the 1/8 model are the same as each other. However, the amount of resources required to solve the full model is much more than the difference of the number of elements. Note that CPU time increases exponentially.

| | Full Model | 1/8 Model |
|---|---|---|
| CPU Time | 200 Sec | 7 Sec |
| Memory | 2.42 e5 kB | 2.31 e4 kB |
| Disk | 1.8 e6 kB | 1.66 e5 kB |







A Step by Step Guide

FEA Fundamentals
Method to Evaluate the Results
Practical Examples

# CATIA V5 FEA Release 21

## A Step by Step Guide

Jaecheol Koh

ISBN 978-1470172824 -4

9 0 0 0 0>

9 781470172824